USER'S GUIDE

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USER'S GUIDE

Welcome to the world of machining and graphical creation!

This User's Guide is the reference manual for the widest configuration of Type3. Depending on the optional modules included in your license, some sections may not apply to your case. They will nevertheless allow you to discover the numerous innovative and powerful functions of Type3.

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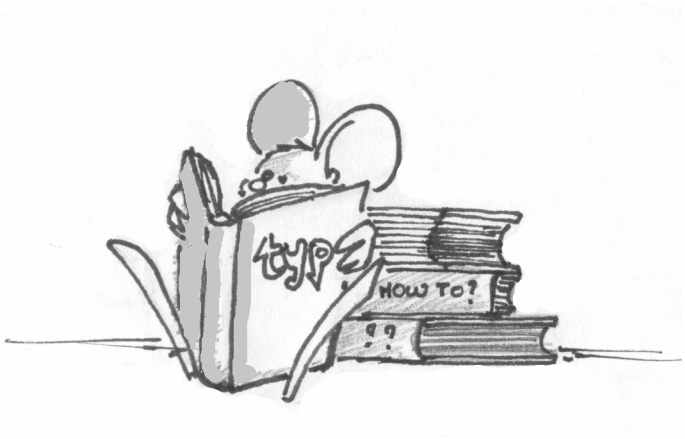
FUNDAMENTALS



OPERATING REQUIREMENTS

Hardware requirements

Microsoft Windows **NT** 4.0 (at least Service pack 6) or Windows **95** (OSR 2), **98** or **2000**.

Minimum required:

1 PC Pentium with

- 64 MB RAM
- one hard disk, 150 MB available (for the fullest license; 250 MB for the full installation)
- CD-ROM driver
- a 17" Windows compatible, SVGA color monitor 800 x 600
- 1 mouse Windows compatible,
- 1 parallel port

Equipment recommended:

- a 21" screen
- 1 video card
- 1 scanner TWAIN compatible
- 96 MB RAM

If you need to free some disk space, you can, once the installation is complete, delete in the **Language** sub-directory the directories corresponding to interface languages not meaningful to you (e.g. the **It** directory for Italian). About 30 MB can be recovered by language.

Machining equipment

Type3 directly drives most of the 2D engraving machines available on the market as well as vinyl cutting or waterjet equipment. For laser cutting and engraving, contact us and receive information about the product **LaserType**.

As an option, **Type3** can access 3D engraving, with the sharpening of angles, with modeling (TypeArt module), and in 3, 4, or 5 axes. Any software from the **Type3** family has post processors to drive most of the 3D engraving machines.

For the 2D engraving machines an adaptation kit for 3D is available: **CNC PC 2** (for more information, please contact us).

INSTALLATION

Attaching the dongle

- Turn your computer off.
- Remove any cable or any other dongle connected to your printer port (LPT1).
- Plug the male side of the dongle into the parallel port of the computer.
- Plug into the back end of the dongle any other cable or other security plugs that were previously connected to the printer port.
- Turn your computer on.
- The printer must always be connected and « On line ».

Installing the software

The software

- Insert the CD-Rom in the CD driver (example: "d" if your CD ROM driver is installed on "d").
- Select **Start**, then **Execute** and the **Browse** button.
- Open the **Type3V42000** directory and double-click on the **Setup** program.
- Follow the instructions of the installation program.

The dongle

The software has been conceived to detect automatically the dongle whatever the system you are working under (Windows 95 / 98 / 2000 or Windows NT).

Nevertheless, in case you had some problems, when the installation is complete

- Select **Start**, then **Execute** and the **Browse** button.
- Open the **C:\Vision\key** directory and double-click on the **Setup** program.
- Drivers are installed though nothing appears on screen. Restart Windows.

For updating the software, you will need to inform us of the codes of the dongle. For this, in the menu ? **of the CAD module, select the option** About... **and click on the function** Order**.**
Print this page after filling in the required information and fax it to us or to your nearest distributor.
We will get back to you the same way. See Appendix 3.

Installing True Type fonts

To use the special fonts, you must first install them in Windows (after complete installation of the software).

Start --> Parameters --> Control Panel, then **Fonts**,
In the **File** menu **--> Install fonts**, select all True Type fonts in **c:\Vision\fonts\ttf** directory.

Choosing the interface language

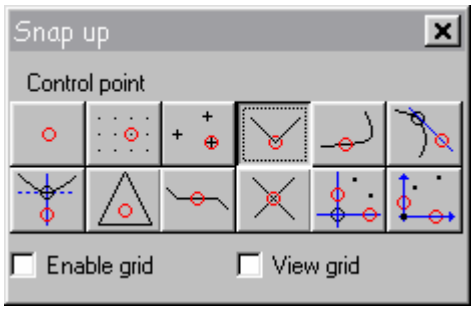
To change the interface language, choose **Options** in the **Edit** menu. Select the **General** tab in the dialog box displayed on the screen and then the desired language in the scroll-down menu.

Click on **OK** to validate. You need to restart the application.

Function keys

Some function keys are used every day in the drawing module, and you should know about them as soon as possible:

F2 : to specify precise numerical coordinates during an operation.
F3 : to select a snap mode when working on several elements (Ctrl-F or Ctrl-D allow to change mode without changing the current tool).



F4 : to preview the transformation in progress (keep pressed).
F10 : to open the **Options** dialog box, from where you change several working parameters.

On-line help

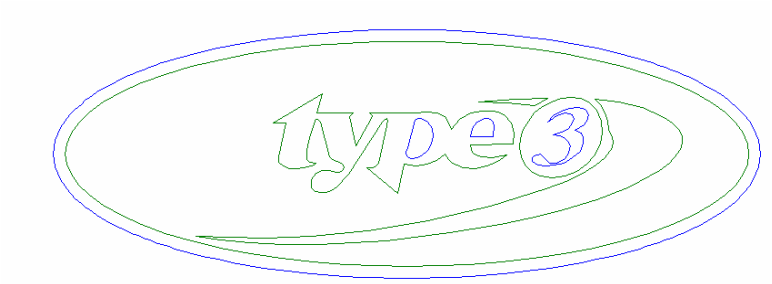
The on-line help is accessed with the **F1** key. Very comprehensive, it follows the structure of this Guide and will provide relevant information during your work. Many active areas in toolbars and screen shots will launch a direct link to related topics.

GETTING STARTED

Getting started

This tutorial takes you step by step through the basic functions and will guide you through the creation of your first machining sample.

You are going to create a 3D cutting from **Type3** logo:



This exercise will take you about 10 or 15 minutes, with the following steps:

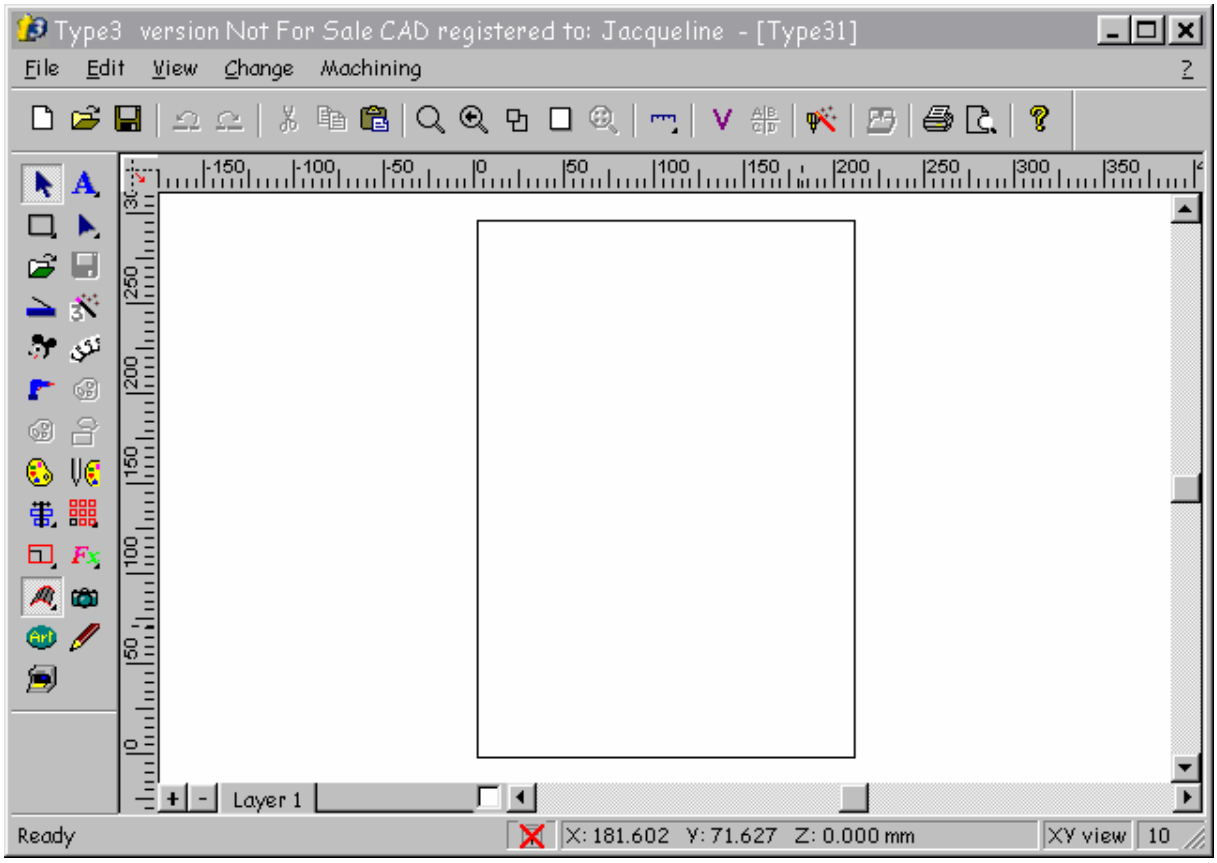
- Opening a document
- Creating a toolpath
- Defining a tool
- Simulating the toolpath
- Configuring the machine
- Control of the output codes
- Preparing the machine
- Launching the machining

Running the program

To launch the software, double-click on the icon:

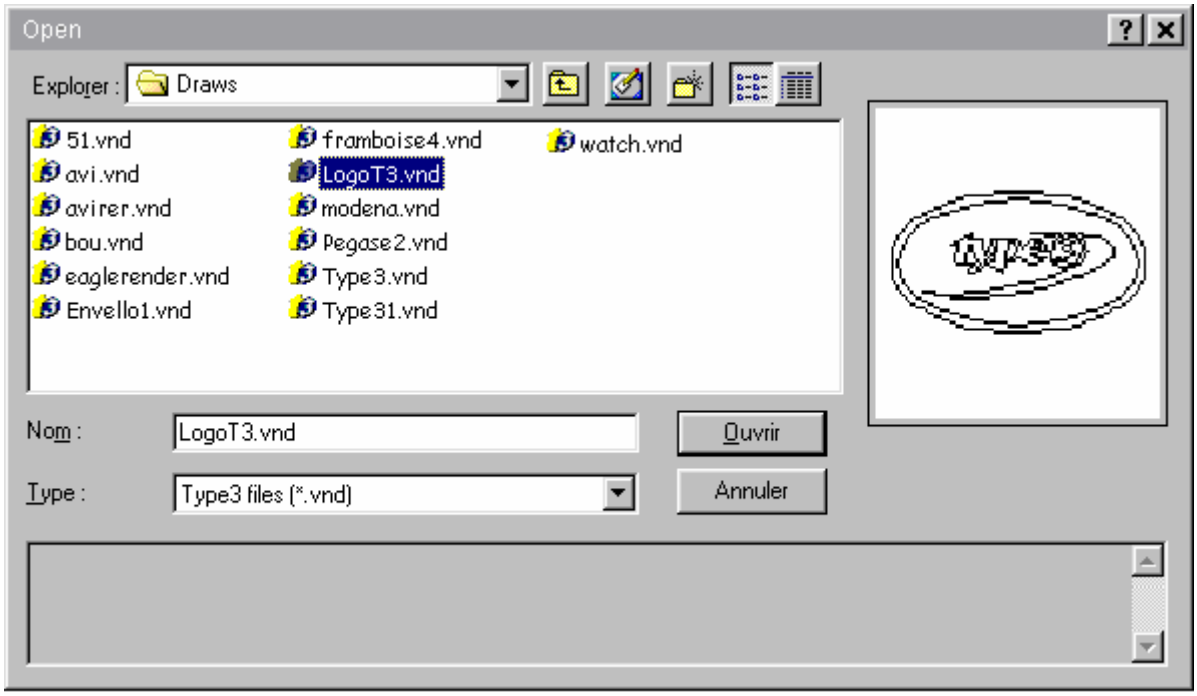
Remark

Unless otherwise stated, clicking and double-clicking are done with the left button of the mouse.


Type3 version Not For Sale CAD registered to: Jacqueline - [Type31]
File Edit View Change Machining
Layer 1
Ready
X: 181.602 Y: 71.627 Z: 0.000 mm
XY view
10

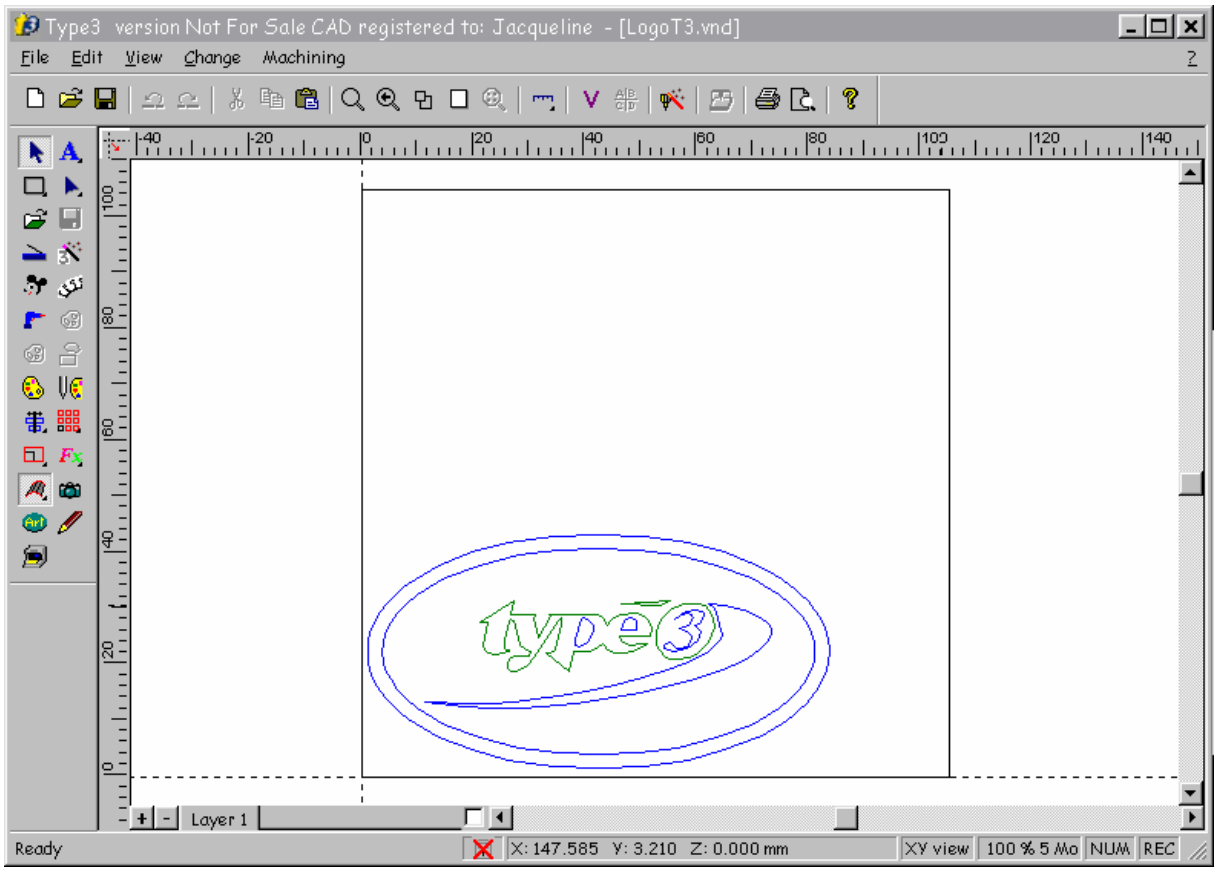
You are now in the **Design Module** (drawing and CAD), also called TypeEdit. From here you can import, create and work on the design from which you will make the engraving or cutting.

Click on the **Open** icon in the upper toolbar to load an existing drawing.



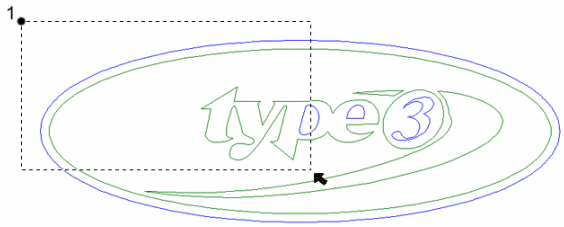
Click on the file called **LogoT3.vnd,** then click on the button **OK**.

The drawing now appears on your screen, but we will not change it.

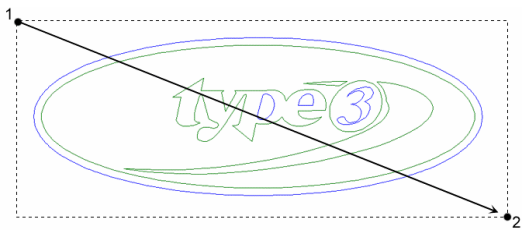


Sending the object to the CAM module

First, you have to select the design you wish to send. Click on a point (point 1 in the illustration) located outside and above the drawing (do not release the mouse button).



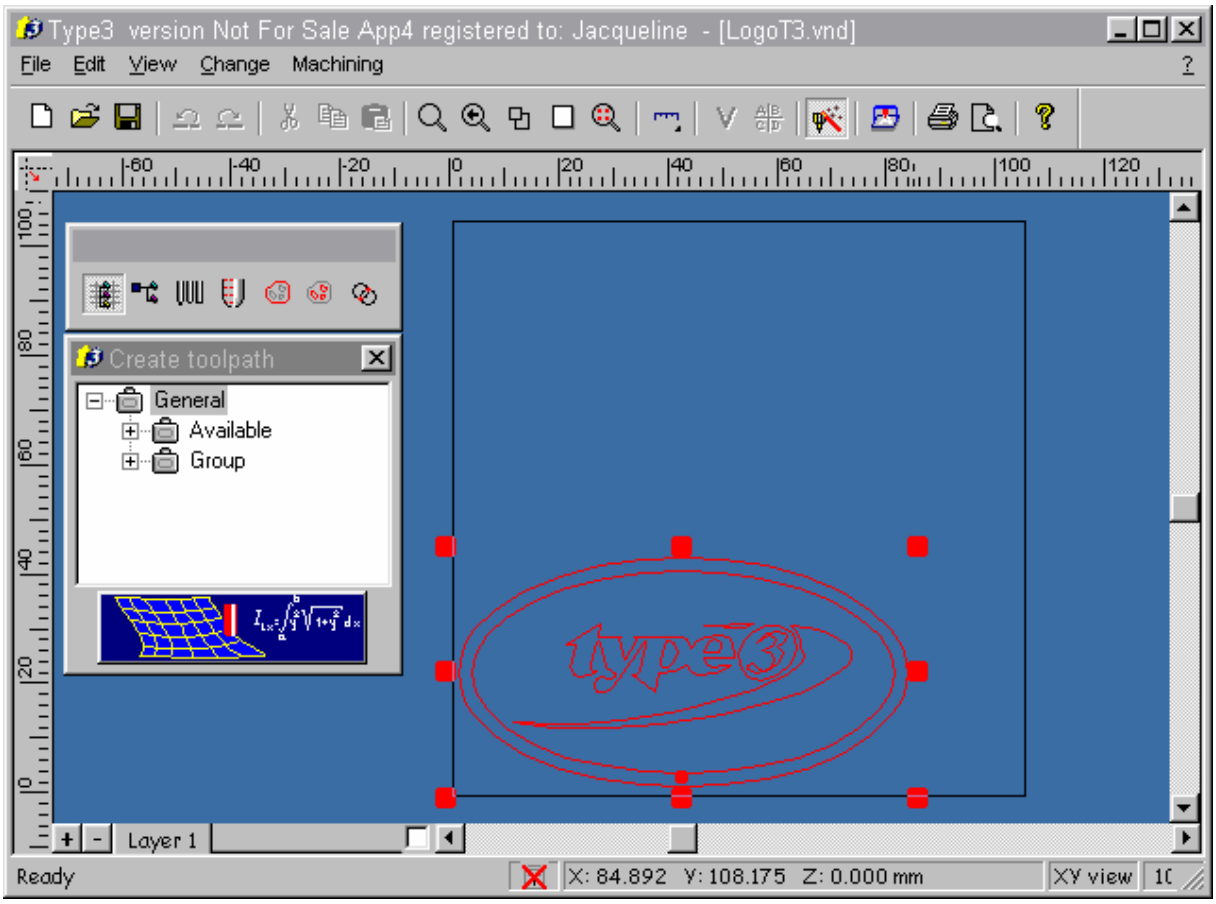
Drag the pointer towards a point 2, outside and below the drawing.



Now release the mouse. Be careful, this logo is made out of two items: the whole drawing must become red!

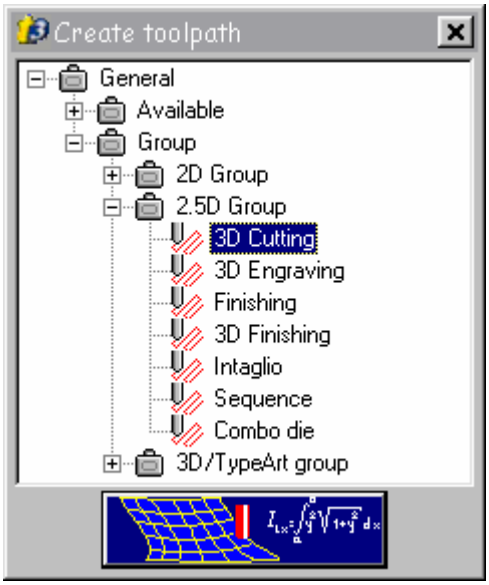
Click on the **CAM module** icon .

You see now the logo selected in the machining module, which has a blue background color. You switch back to the CAD module using the same icon.



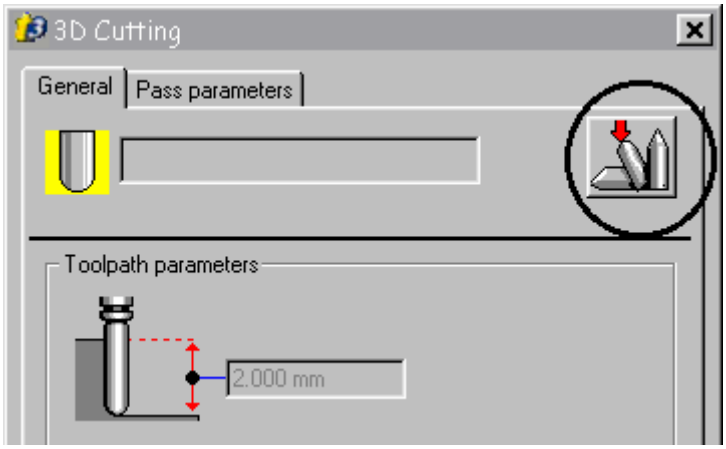
Creating a toolpath

Click on the **Create toolpath** icon located in the toolbar of the **CAM module**. The **Create toolpath** dialog box is displayed.



In the **Create toolpath** window, double-click on **General**, then on **Group** and finally on **2.5D Group** to access to 3D Cutting toolpath. Double-click to select it: the corresponding dialog box is displayed on the screen.

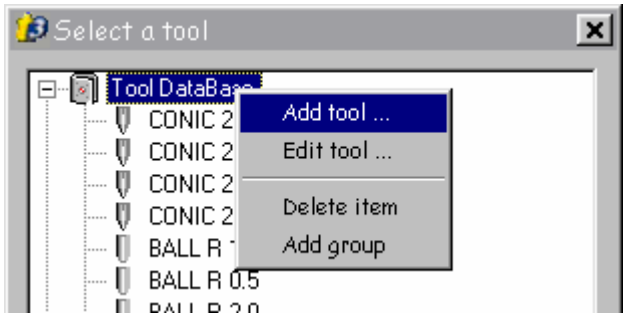
Note: maybe your software license does not include the 3D Cutting toolpath. Choose then 2D Cutting in the 2D Group: apart from true angles, the toolpath and its parameters are very similar, and you'll be able to perform entirely this Getting started section.



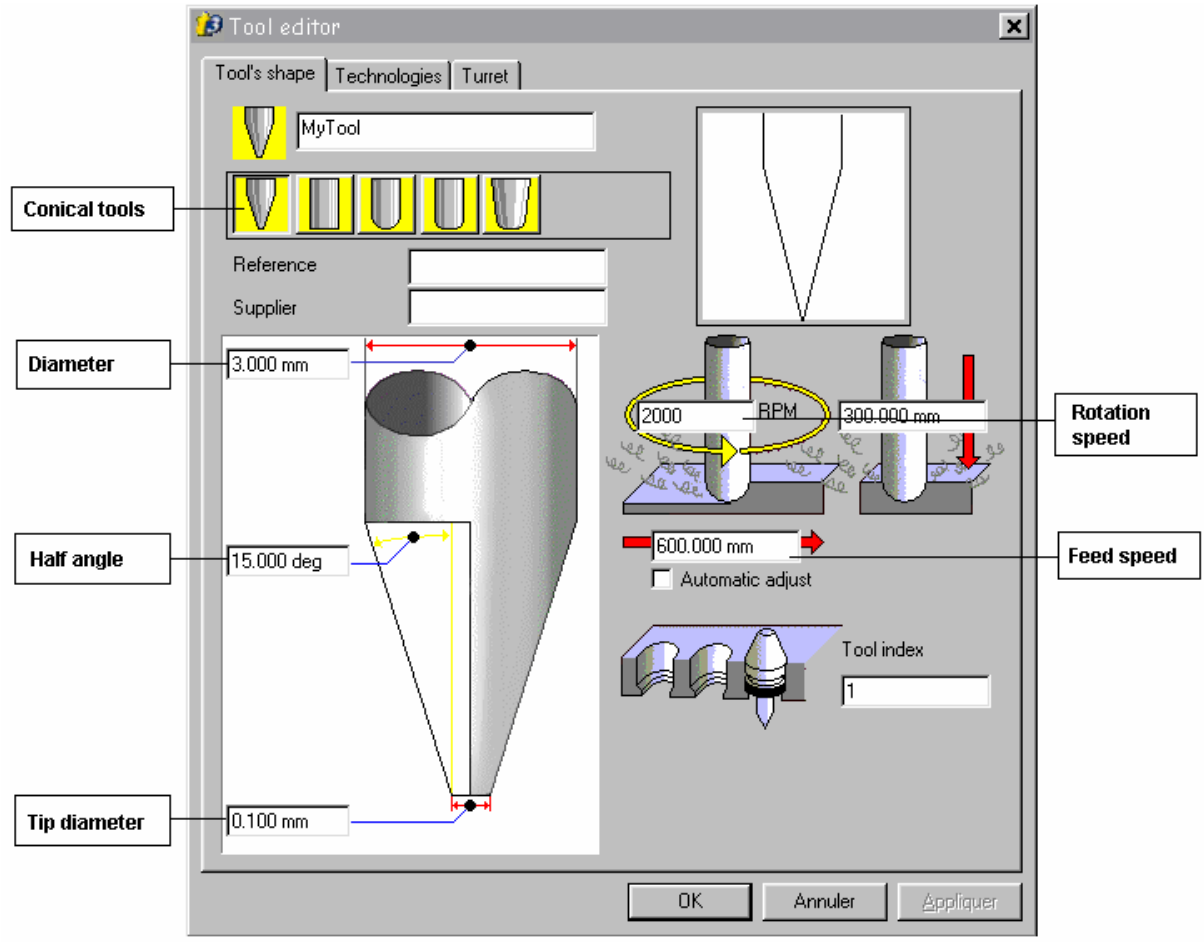
For each machining, you must define a tool and a toolpath, in other words a machining method. We will specify the settings of the tool used for the 3D Cutting.

Creating a conical tool

To create a tool, click on the icon in the 3D Cutting dialog box.



The **Select a tool** dialog box is displayed on the screen. Select **Tool Database** in this window and click on the mouse right button. In the displayed context menu, click on **Add a tool…**. The dialog box **Tool editor** appears.



To add a tool in the tool database, you have to name it (**MyTool** in our example), select its geometry, set its parameters and save it.

- Diameter: 3.000 mm
- 1/2 angle: 15.000°
- Tip diameter: 0.100 mm
- Nominal feed rate: 600.000 mm/min
- Rotation speed: 2000 RPM

Choosing the depth of cutting

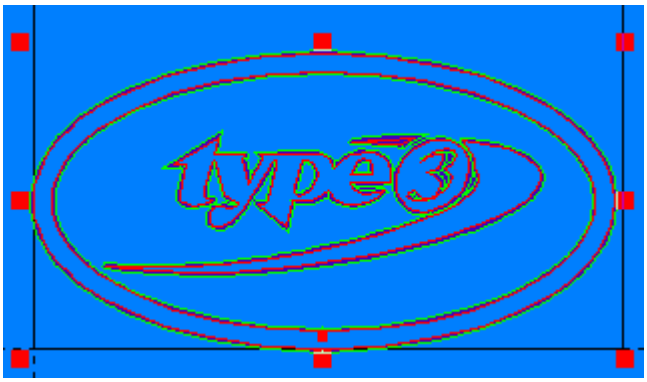
Still in the tool editor, in the field **Depth of cut** situated in the **Technologies tab**, enter a value of 2 mm.

Once all the desired settings are done, validate the choice of your tool by clicking on **OK**.

Remark

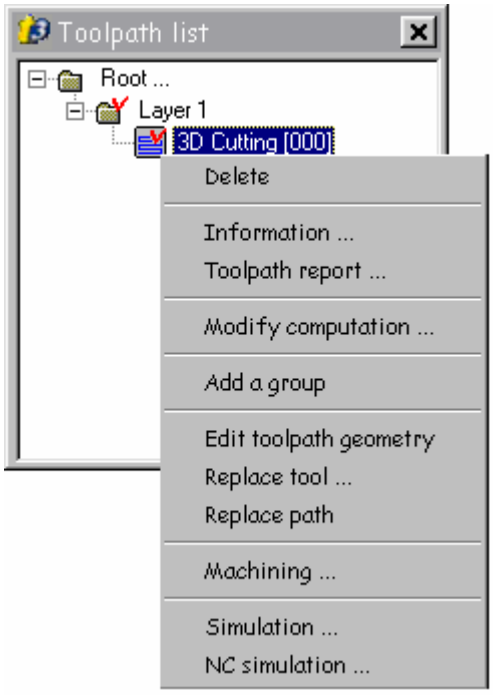
The created tool is automatically saved once you've clicked on OK **in the** Select a tool **dialog box (after having closed the** Tool editor **through OK). Then you can use this tool as many times as you want.**

All dialog boxes being closed, the software calculates the exact toolpath in 3D. It is gradually displayed on the screen with a color different than the drawing.

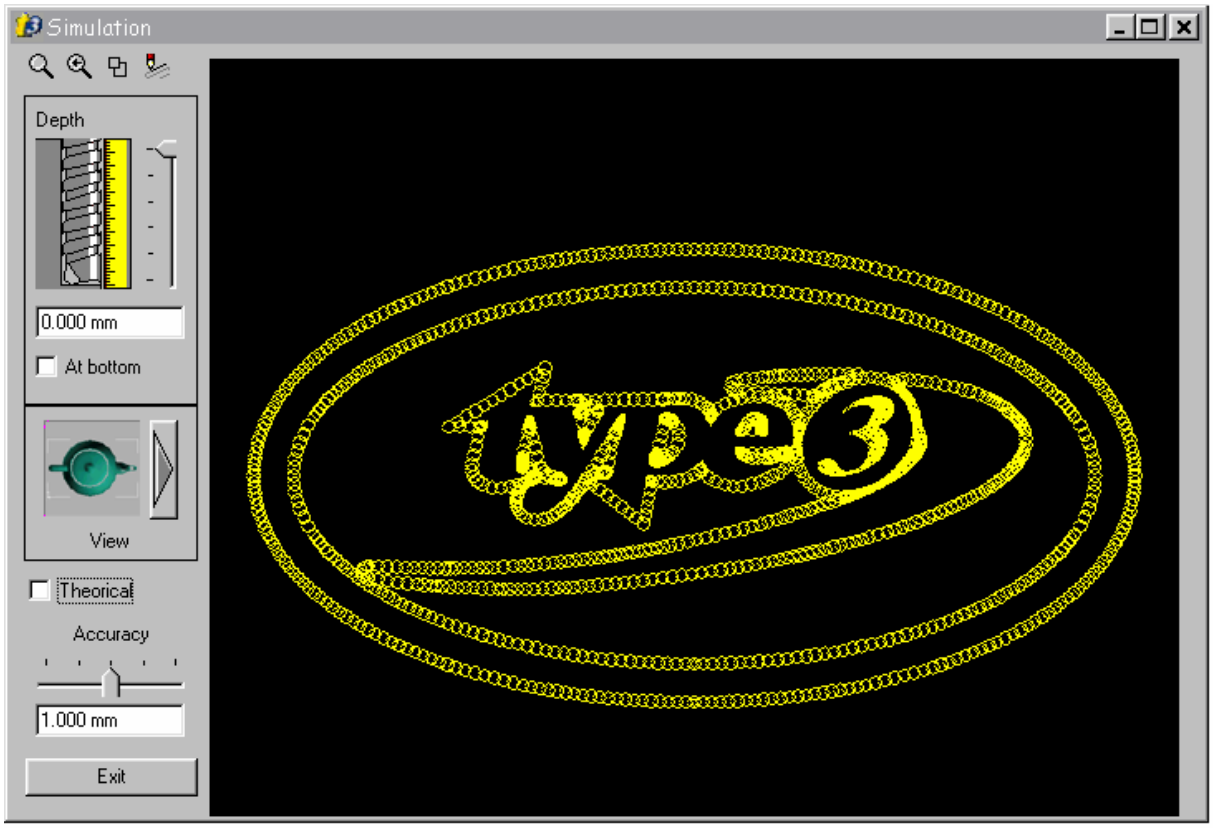


Simulating the toolpath

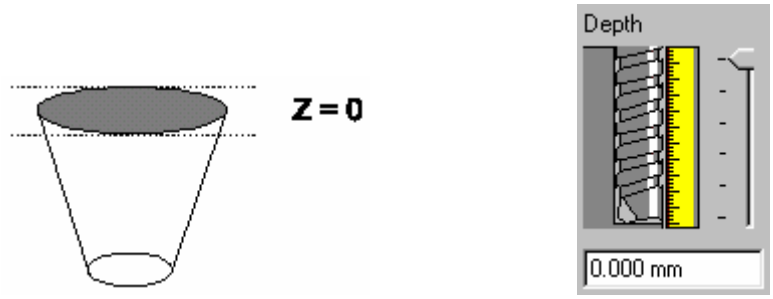
When the toolpath is generated, it is ready to be sent to the machine. But before, we will visualize it realistically. Click on the **Toolpath list** icon of the CAM module toolbar. The corresponding dialog box opens.



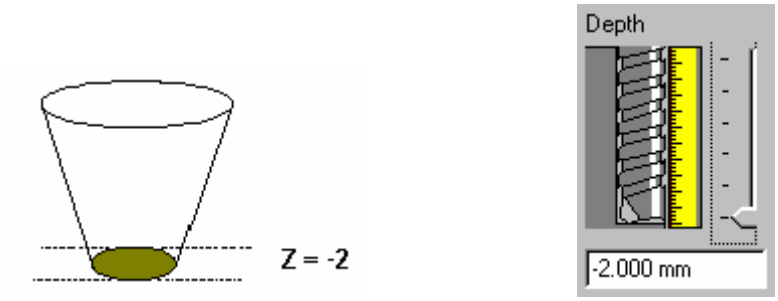
Select the toolpath you want to visualize, and click the right mouse right button to display the context menu. Click on **Simulation…** in the context menu. The corresponding window opens.



By default, the simulation window displays the created toolpath on the top of the cutting (the machined plate). In our example, the height is Z = 0 mm.

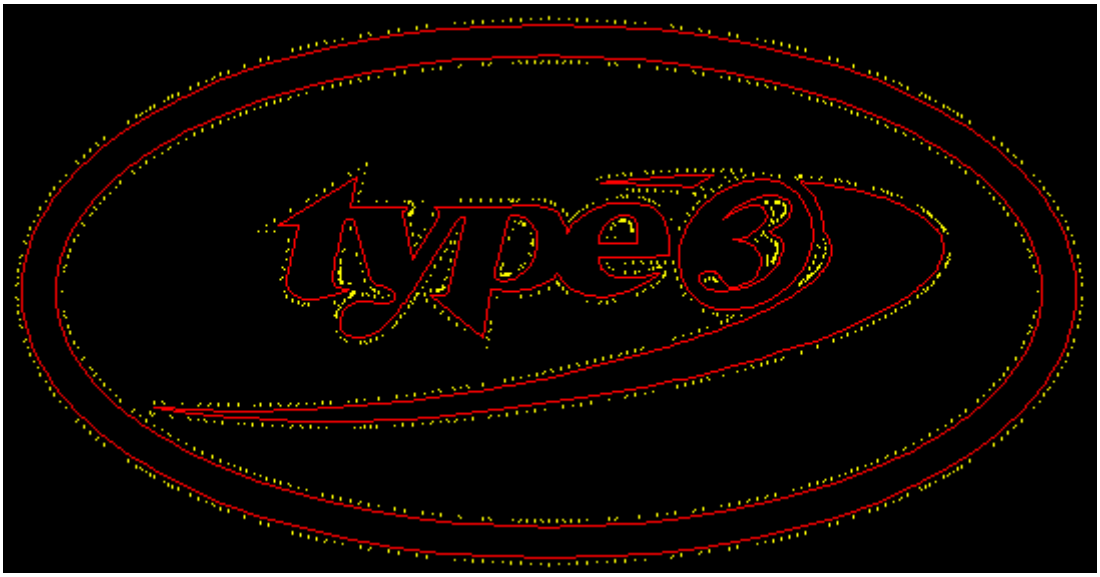


Section of the tool at Z = 0



Section of the tool at Z = -2

Nevertheless, you can modify the visualization height. Use therefore the view depth's cursor or enter the desired value in the numerical field located below.

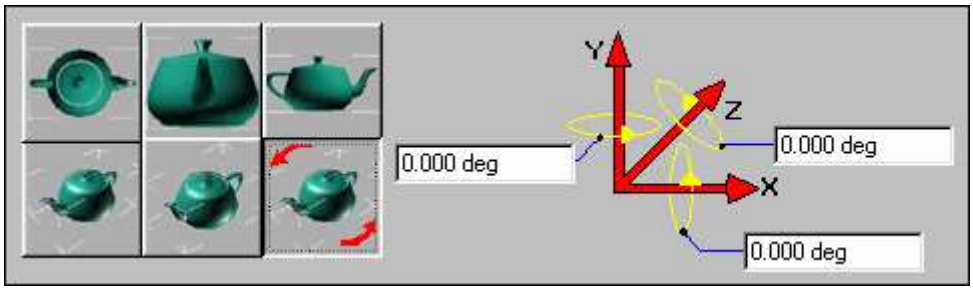


Visualization at the bottom of the cutting

Checking the **Theoretical** box, the theoretical contour of the drawing is shown in red color, allowing you to compare the machining to the original design.

Viewing angle in the simulation module

To modify the viewing angle of the toolpath simulation, click on the icon of the dialog box **Simulation**. The toolbar of the view options is displayed on the screen.



Double-click on the desired option. The visualization will be automatically refreshed.

Remark

If you choose like in the example shown above the Free 3D view **(symbolized by the icon with arrows), you must enter the values that define the viewing angle in the corresponding numerical fields.**

Zooming

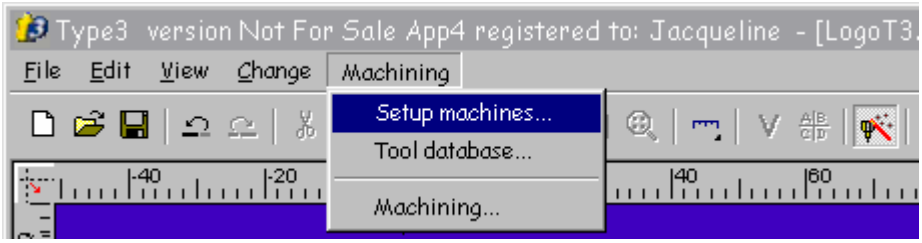
You can enlarge the drawing by positioning the cursor on the drawing and by clicking on the right button of the mouse. You can also click on the icon. To go back to the original view, you can either click on the icon with the mouse left button, or double-click with the right button.

Leave the simulation window using the **Exit** button.

Sending the cutting to the machine

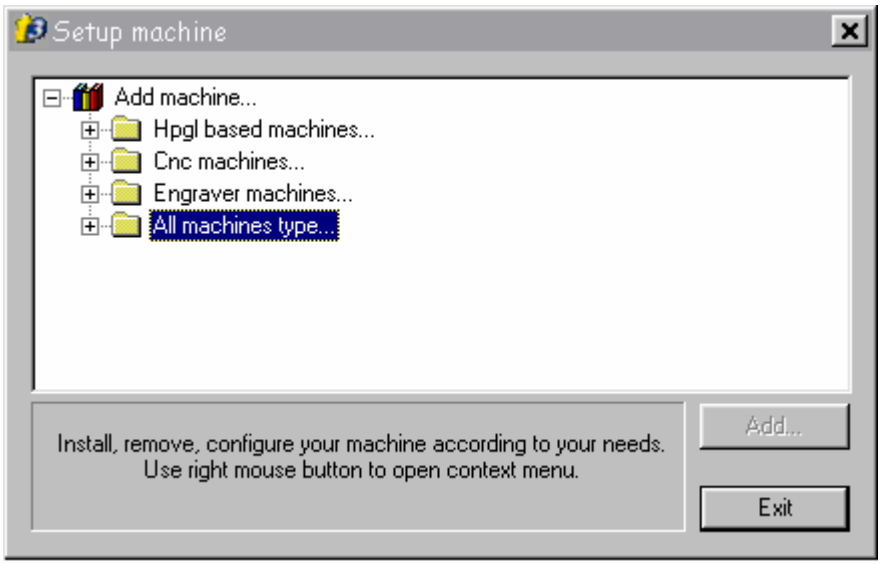
Setting up the machine

Before sending the cutting to your machine, you need to install the machine itself!



Select therefore **Setup machines…** in the **Machining** menu.

The **Setup machine** dialog box is displayed on the screen.



Double-click on the **Add machine…**.folder.

Select your machine among the different lists of machines available. If you do not know the type of your machine, select the **Demo machine** in the **All machines type** list.

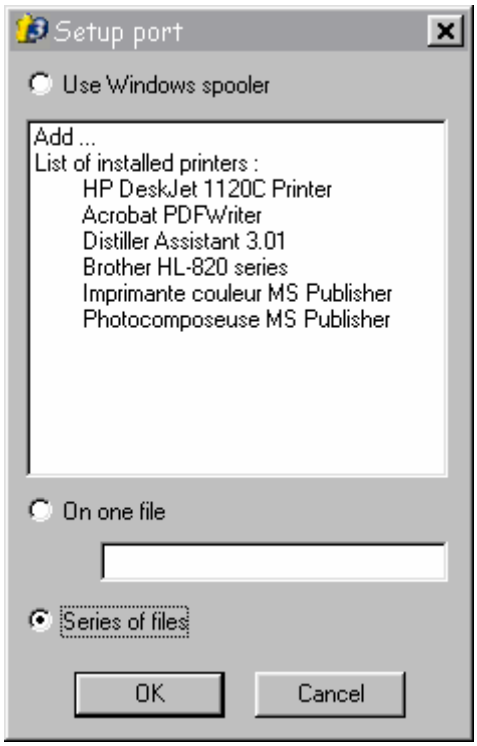
Click then on the **Add…** button to register your selection. The dialog box **Setup port** is displayed on the screen.

Choosing the port

If your machine is actually plugged to the computer, select the option **Use Windows spooler** in the dialog box **Setup port** and click on **OK**.

If your machine is not connected to the computer you are using, you will have to make a machining file and copy this file on a floppy disk, or over a network.

The default name of the file generated with the option **Series of files** is the name of the document with the extension "Uxx" (in our case "LogoT3.U00"), in the same directory as your drawing (i.e. \Vision\Draws).



Select the option **Series of files** of the dialog box **Setup port** and click on **OK** to validate your choice.

Machine parameters

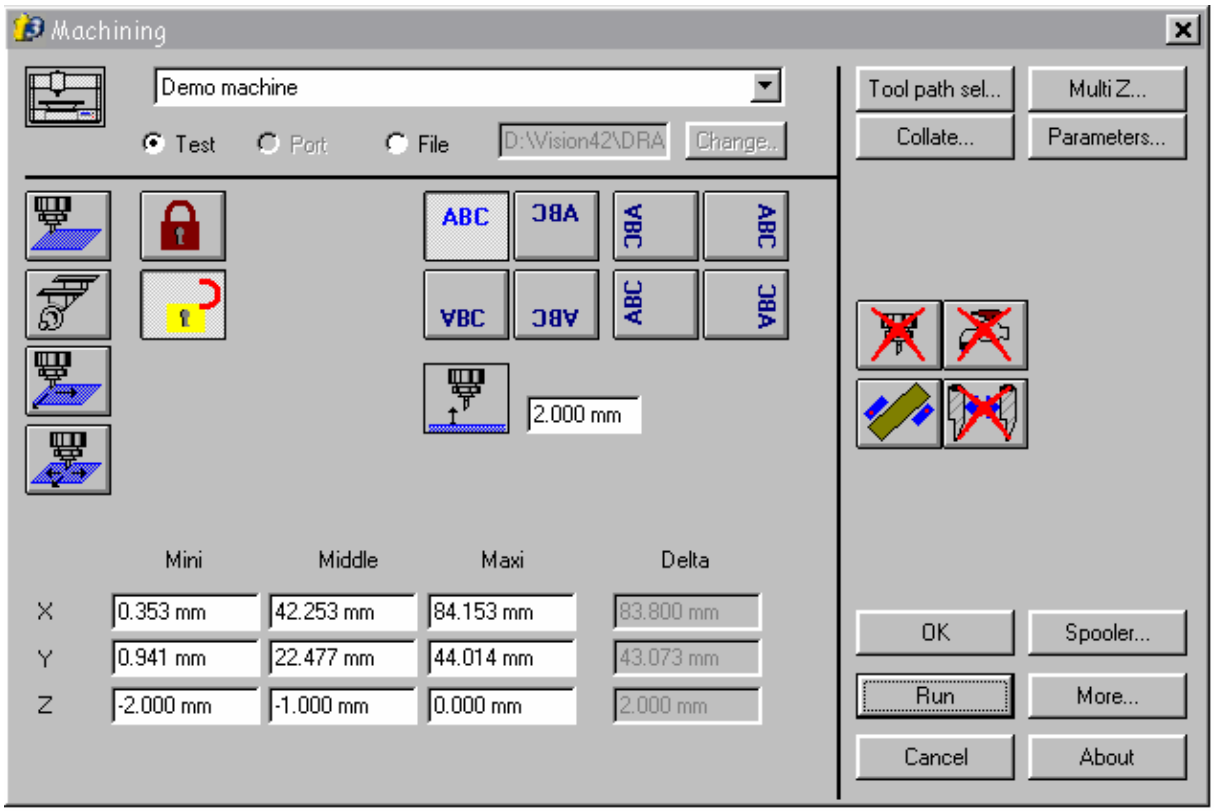
For now, just validate the default parameters of the second **Setup machine** dialog box that appears once you have clicked on **OK** in the **Setup port** box.

Once you've completed all these operations, click on the **Exit** button to go back to the CAM module.

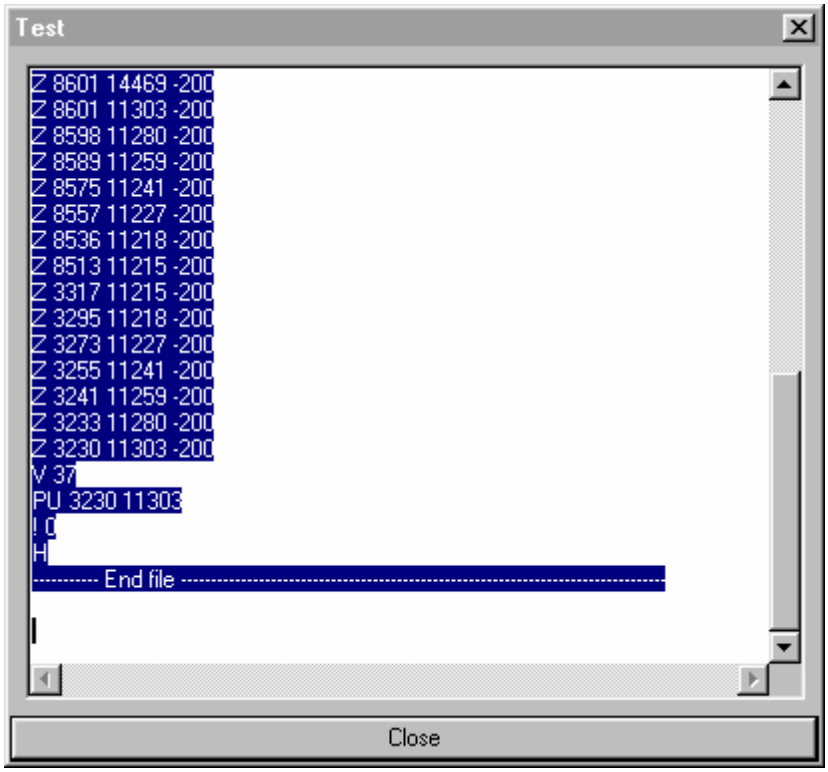
Verifying the output codes

In order to display the instructions sent to your machine, select the toolpath in the **Toolpath list** dialog box and click on the right button. In the context menu, choose the option **Machining….**

The **Machining** dialog box is displayed on the screen.



Select the **Test** option in the upper part of the dialog box and click then on the **Run** button. The corresponding dialog box is displayed on your screen.



This dialog box displays what will be sent to the machine. Finally click on the lower bar **Close** to exit.

Machining

You can now try out your machine. Again click with the right button on your toolpath in the **Toolpath list** dialog box. In the context menu, select **Machining…**. To display again the corresponding dialog box.

You must now select:

- **Port**: if your machine is connected to your computer and if you have set up the printer connection.
- **File**: if you want to create a machining file and transfer it to the machine later.

Click on the **Run** button in order to output the codes, and finally click on **OK**.

Before starting the machine

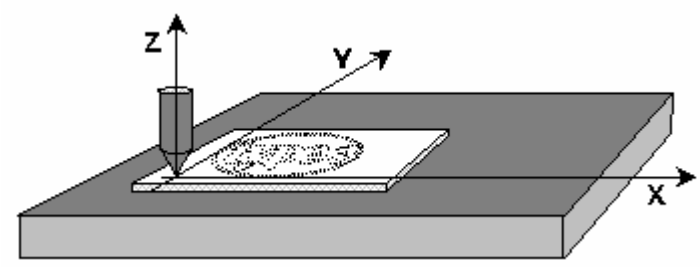
Make sure you have installed on the machine itself the same tool that you have created in the **CAM module** during the previous steps.

Remark

The half angle and the tip diameter are very important! Always check that you are using exactly the same tool on your machine as the one that has been used for the toolpath calculation.

Place your material with double side adhesive tape to a bigger plate, which is fastened onto the table of your machine (so you won't damage your engraving table).

After that, check your X,Y zero point and Z zero point on the surface of the material.
(Z = 0 on the surface of the material when carefully positioning the tool in contact with it).



You can now run your machine.

Congratulations! **You have just created your first cutting of a logo in 3D!!!**

How to return to the CAD module

Click on the icon to go back to the **CAD module**.

We recommend you now to read the next sections of this Guide, and thus discover many other functions of the software.

CONFIGURATION

Plate dimensions

To modify the basic configuration and to parameterize the dimension of the plate, click on **Material** in the **Change** menu and enter the new value in the corresponding fields of the **Material definition** dialog box.

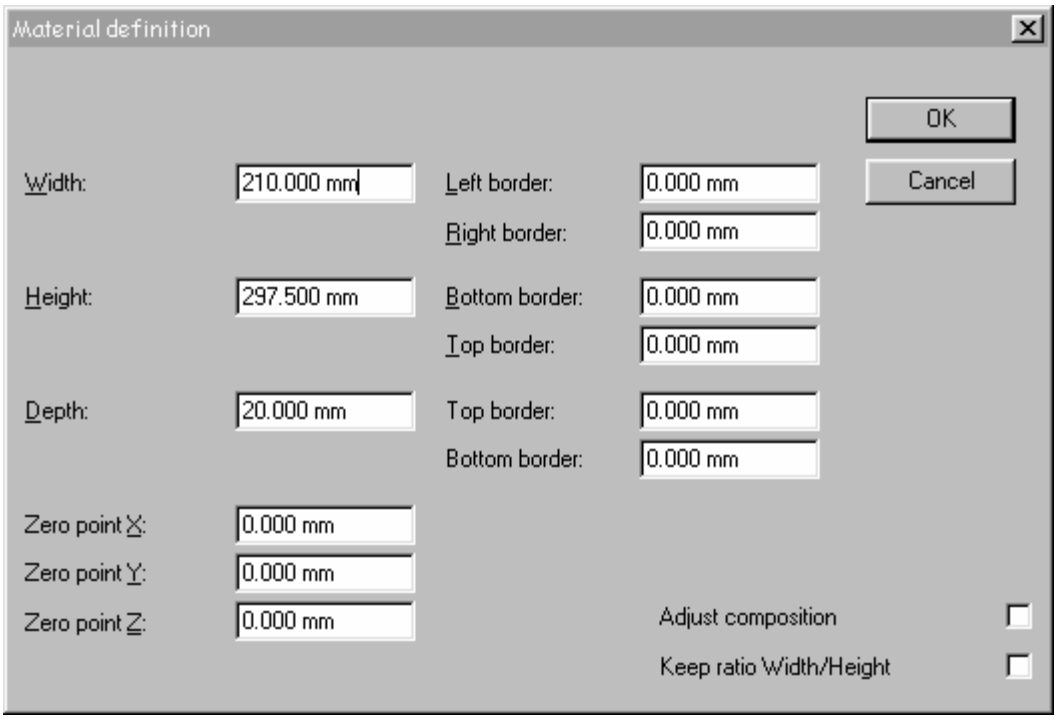


Plate dimensions
The size of the plate is modifiable according to your needs. To modify its dimensions, click in the menu **Change → Material**, and enter in the fields **Width**, **Height**, and **Depth** the desired values.

Borders
Borders are defined in the **Material definition** dialog box under the **Change** menu.
The surpassing of an object outside the selected area will activate a warning message before engraving.

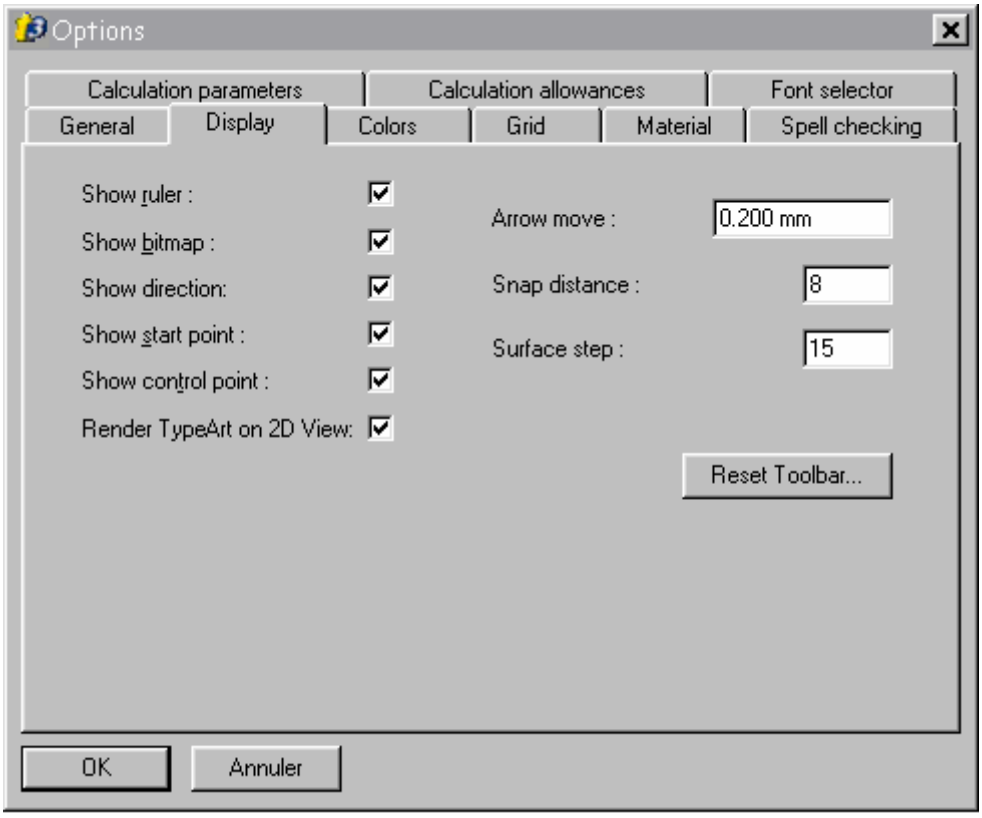
Zero point
It defines the origin of the reference points X, Y, and Z on the active surface.
To modify it, enter the corresponding desired values in the fields **Zero point X**, **Y**, and **Z**.

Adjust composition
When the size of the active area has been modified, this option will automatically adjust in accordance to the new size of the composition. For instance, if the desired size is twice as large (and the height remains unchanged), the dimensions of the object will be doubled in length.

Keep ratio Width / Height
When the size of the active area has been modified (non proportional modification), this option maintains the geometry of the object without deforming it.

Displaying options

To modify the basic configuration, chose **Options…** in the **Edit** menu (or hit the **F10** key) and select the displaying options in the **Display** tab, in accordance with your needs.



Show rulers

To display the horizontal or vertical rulers as a reference for positioning.
Hit **F10** and check the option **Show ruler**.

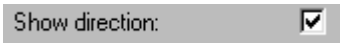
Show bitmap

To display the bitmap pictures (TIFF, BMP, etc.).
Hit **F10** and check the option **Show bitmap**.

If this option is not selected, a crossed rectangle will represent the bitmap picture.

Show direction

To display in color the direction of rotation of the contours.
Hit **F10** and check the option **Show direction**.



By default, clockwise contours are blue, and anti-clockwise contours are green. The direction of the contours is important for machining, filling in, or cutting.

Show starting point

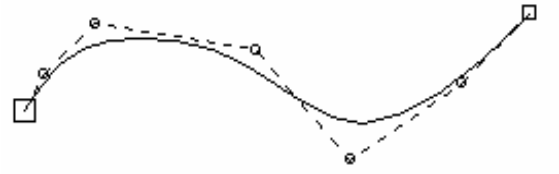
To display the starting point of the contours (indicated by a square).
Hit **F10** and check the option **Show start point**.

In machining, this point will be the starting point of the tool.

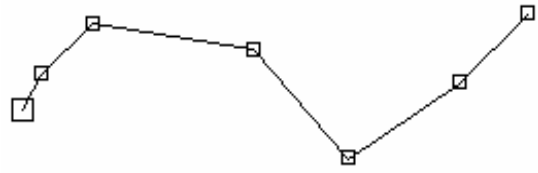
Show control point

To display the control points of the contours.
Hit **F10** and check the option **Show control point**.

For the points out of the curve:



For the points on the curve:



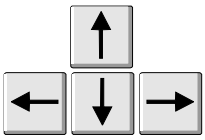
Remark

In some cases, the points on the curve can be indicated by circles. This is especially the case when points have been manually added on an existing tracing.

Render TypeArt in 2D view

Activate this option to realistically render TypeArt objects in 3D but also in 2D because its is often much easier to work with a 2D view.

Arrow move



To modify the value of one move each time you move the selected item with the keyboard arrow keys.
Hit **F10** and enter the desired value in the **Arrow move** field as shown in the example below.

Snap distance

This value corresponds to the snap distance of the contours while using the **Snap** mode initiated by the F3 key.
Hit **F10** and enter the desired snap value in the **Snap distance** field.

Remark

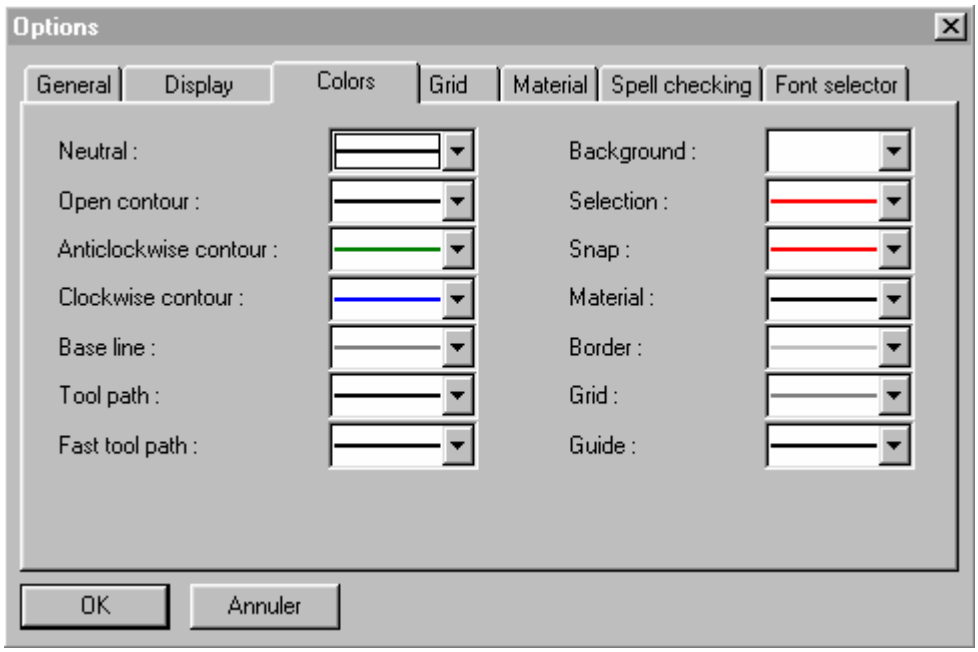
The higher the value, the more limited the snap distance will be. The distance is given in pixels.

Surface step

This parameter defines the accuracy of the display concerning imported 3D shapes, in other words, the number of isometrics for the display of surfaces of revolution or surfaces described in **IGES** format.
Hit **F10** and enter the desired value in the **Surface step** field.

Displaying direction

Select **Options…** in the **Edit** menu. Choose the **Colors** tab in the **Options** dialog box. This displaying mode allows one to recognize the direction of rotation of contours: blue for clockwise contours, green for anti-clockwise contours.



The modification of this parameter will affect the milling process.

Displaying the wire contours

In the **View** menu select the option **Wire color.**
By default, the objects appear in black wire contour. The color of the contour can be modified.

Select the object, click on the icon **Colors** , and double-click on the desired color to validate.

Displaying the contours in fill color

Colors
Select in the **View** menu the option **Fill color**.
By default, the objects appear in black wire contour. The color of the contour can be modified.

Select the object, click on the icon **Colors** and double-click on the desired color to validate.

Remark

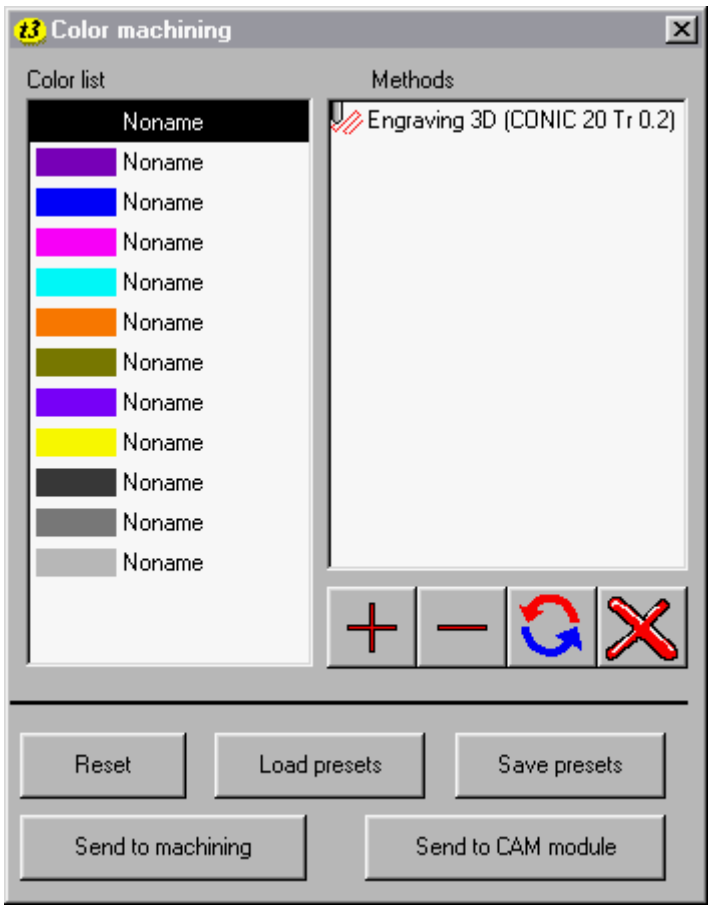
Visualizing the recess or the relief of a design becomes interesting when using this mode with the Combine function.
The recessed areas in machining are displayed in color.

Machining with colors

This feature allows you to assign a kind of machining to a color defined during the engraving's creation.

Previously, you must assign a color to each contour thanks to the **Color** dialog box which is displayed on the screen by clicking on the icon.

Click then on the **Colors for CAM** icon . The **Color machining** dialog box is displayed on the screen. You can now create a *Color/Toolpath* combination.



Click on the icon to define a toolpath for the selected color.

Click on the icon to delete a created toolpath for the selected color.

Once a toolpath is associated with a color, you can modify the name of this color by clicking on it with the right button.

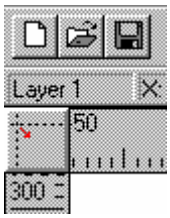
In this way you can define in advance the whole machining sequence, changing the order of the color machining (drag and drop with the mouse). The same color can appear several times in the same sequence at different moment of the machining.

When you click on **Send to machining**, all defined toolpaths are launched following the specified order thanks to the associated colors.
Click on the **Send to CAM module** button so that you create the toolpaths in the CAM module to have the possibility to access it later and send to machining.

You can also save a group of combinations by clicking on the **Save presets** button or load existing combinations by clicking on the **Load presets** button.

Modifying the origin of axis

You can modify the point 0 of the axes, which is by default at the bottom left hand corner of the active surface.



Click on the button, drag the mouse, and release the button when you have reached the desired point.

Automatic saving

To launch the automatic save mode for current projects, go to the menu **Edit → Options…**, choose the **General** tab in the dialog box **Options** and enter the number of minutes desired in the **Auto save** field.

MANAGING FILES

Creating a new file

New file
You can create a new file with this icon or through the **File** menu, and **New**.

By default, the new created file is labeled [UNTITLED]. This name is automatically attributed to any new file. When you are creating a new document, we advise you to save it under a new name since the UNTITLED file does not save the tool paths.

Opening an existing file

Open a file
You can open an existing file with this icon or through the **File** menu, and **Open**.

Select in the dialog box:

- the drive,
- the directory where the file is located,
- the name of the file, and validate.

Saving a file

Save
You can save your drawing at anytime. If the file already exists, you can execute an automatic save; if it is a new file, you need to name it in the **Save as** dialog box.

Select in the dialog box:

- the drive,
- the directory where you want to save your file,
- enter the name of the file, and validate.

Information

To find a drawing more easily, you can attach to it key words or other information such as a creator's name, a customer's name, or any other comments.

Under **File → Information**, enter the desired information and click on **OK**.

Then save the file as described above.

Remark
To go to the next line within the commentary section, press the Ctrl **and** Enter **keys at the same time.**

Opening a model

For repetitive works, a library with pre-registered plate models is at your disposal. You can complete it at any time.
Besides the dimensions of the plate, a model can contain a text, a logo, variables, etc.
To open a model, select under the **File** menu the option **Open model**.

In the displayed window, double-click on the selected model.

Saving a model

Enter the corresponding parameters for the desired plate and select the option **Save model** under the **File** menu.

Under the window **Save as**, enter the name of the model in the **Name** field and click on **Save**.

By default, the directory selected for the saved work is **Models**.

Importing a file

Import
You can import a file issued from other applications with the following formats:
Bitmap *.bmp, *.Tif / Autocad *.dxf, / PostScript *.eps, / Adobe Illustrator *.ai, / Plotters *.hpg, *.plt, / IGES *.igs, / ISO *.iso / Tool path *u?? / Window Metafile *wmf, JPEG *.jpg.

This function can also be reached through the **File** menu, and **Import**.

Select in the **File import** dialog box: the directory, the name of the drives (A, B, C etc.), the format, the file name, and then validate.

Exporting a file

Export
This function is available in the menu **File → Export**.

You can also export a file in several formats:
Autocad 2D and 3D *.dxf, / PostScript *.eps, / Adobe Illustrator *.ai, / Plotters *.hpg, *.plt, / ISO *.iso, / Symbols *.syb, / Tool path *.u?? / STL ASCII Files ou STL Binary Files *.stl,
to transmit them to other applications, completing the symbol library, for instance.

Choose the directory where you want to save the file, then select the format (dxf, eps, ai, etc.) in the scroll down list. Enter the file name in the field **File name**, and validate.

Finding a file

To find a drawing you can ask for a list of files sorted by either the name of the drawing, by the date of creation or modification, or by the creator's or customer's name.

Click on the **Find** command under the **File** menu and select the searching parameters (name, date of creation or modification...), then choose the document on the screen.

You also have at your disposal a general preview of the selected drawing.

Copying

This function copies the document selected and all the associated files (tool paths, theoretical paths, machining paths, etc.) in the designated directory.

Moving

This function displaces the document selected and all the associated files (tool paths, theoretical paths, machining paths, etc.) in the designated directory.

Erasing

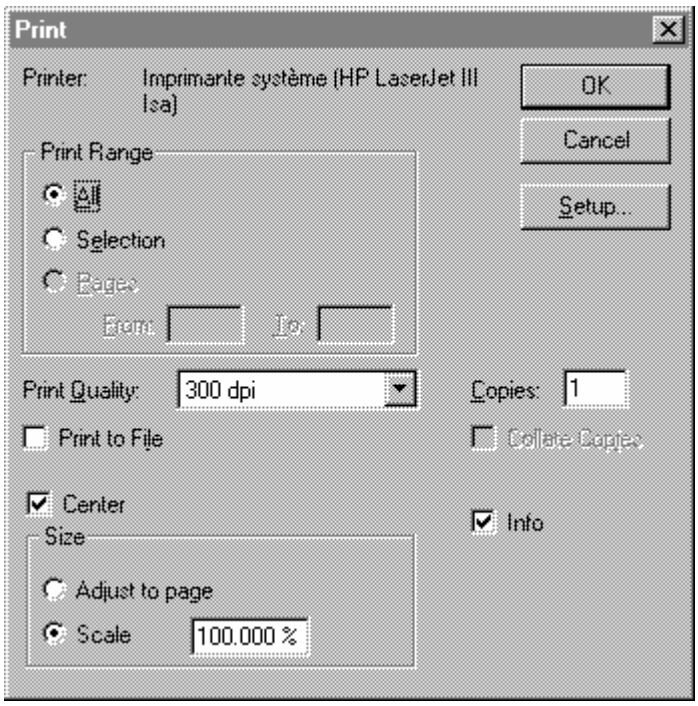
This function erases the document selected and all the associated files (tool paths, theoretical paths, machining paths, etc.)

PRINTING

Preview

Preview
Before printing you can verify whether the parameters of the printer correspond to the desired effect. Click on the shown icon or choose **Preview…** under the **File** menu.

Printing parameters



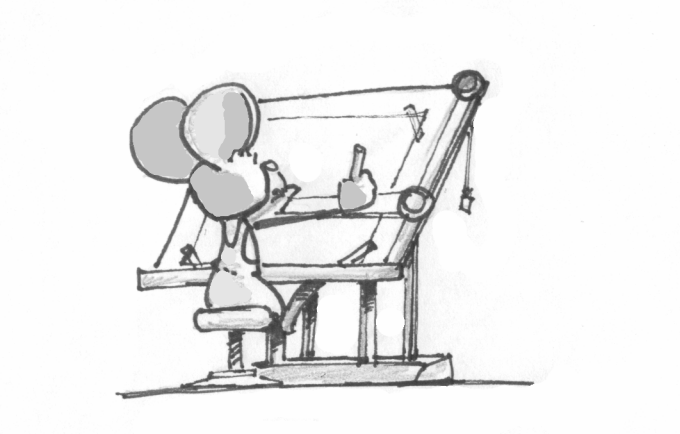
All
To print the whole document.
Selection
To print only a selected area.
Pages
If the print job requires more than one page, enter the page number which you want to print (if you do not want to print the whole document).
Print to file
To send the data to a file so that you can print it later, choose this option (the extension will be ***.prn***).
Center
The drawing will be centered on the paper.
Adjust to page
The size of the drawing will be adjusted to the size of the paper.
Scale
The size will be done according to the scale you enter.
This function is used especially for the creation of chemical engravings of stamps.

Info
To print the information about the document at the bottom of the paper (directory, complete name, size, printing scale, etc.).

Printing

Print
To print the drawing with the selected parameters.

CAD MODULE



DISPLAYING TOOLS

Tools for visualization

The tools concerning the display of an object are always shown at the top of the screen.

You can also find these functions under the **View** menu.

Zoom
Click on the icon and define with the cursor the area you want to enlarge. If the zoom is not enough, redo the operation.
To cancel, click on the same point.

Previous zoom
To jump to the previous view.

Zoom maximum
To display all objects, even if some of them are outside of the active area.

Zoom on active area
To display automatically the entire active area.

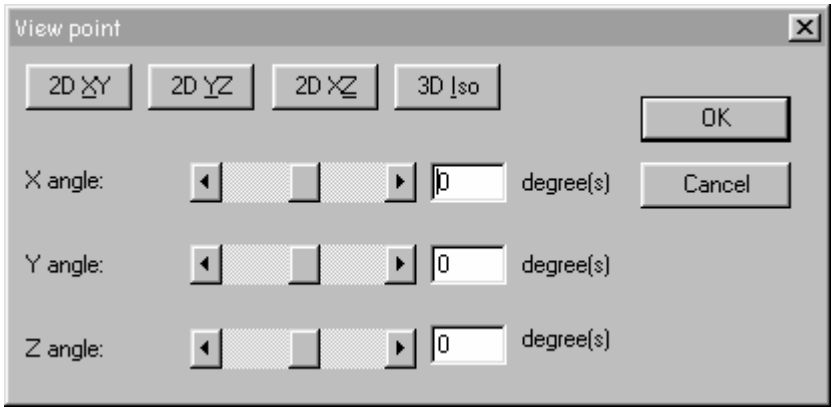
Zoom on selection
To zoom automatically on the selected object.
If there is no object selected, this function is not valid.

Quick tip...
Clicking on the right button of the mouse will enlarge the view (x 2), without selecting an icon.
Double-clicking will give you the maximum zoom.

3D display

The selection of the views can be done either under the **View** menu or by clicking on the lower status bar in the **XY View point** field.
4 types of views are available:

- on the XY axes in front view
- on the YZ axes in cross section view
- on the XZ axes in cross section view
- on the XYZ axes in **3D Iso** view.



You can also use the following shortcuts to select the available views:

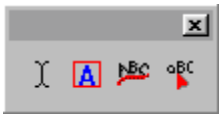
- **Ctrl** + **1**: front view on the **XY** axes
- **Ctrl** + **2**: cross section view on the **YZ** axes
- **Ctrl** + **3**: cross section view on the **XZ** axes
- **Ctrl** + **4**: **3D Engraver** view on the **XYZ** axes
- **Ctrl** + **5**: **3D Iso** view on the **XYZ** axes
- **Ctrl** + **6**: **Free 3D** view on the **XYZ** axes through numerical values (dialog box **View point**).

CREATING TEXTS

Composition tools

Text mode

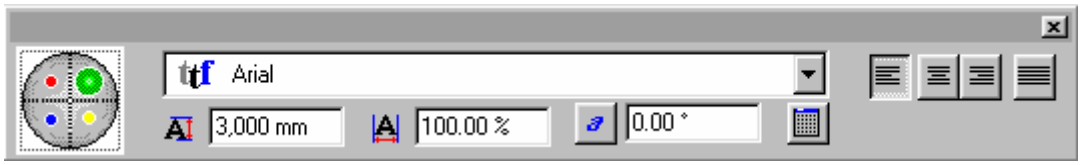
Click on the icon **Text mode** to display the toolbox for the creation of texts.



Description of the selector: Rapido

Free text

To create free text, click on the icon **Free text** which is located in the toolbox for creation of texts. The **Rapido** selector is displayed on the screen.



Menu selection

The circular selector on the left side of the screen, named **Rapido**, allows you to access quickly, with a simple click on one color, one of several menus: layout, styles, character and line attributes.

Validation of the parameters

Click on the left button of your mouse, with the cursor on the active surface, to validate the parameters.
You can also validate your choices by pressing **Enter**.

- ***Transforming automatically into uppercase letters / Automatic Caps***

Uppercase icon **Small caps icon**
Select the text to be capitalized / capitalized in a smaller format, click on the icon, and validate.

Example

Lowercase letters *Uppercase letters* *Small caps*

- ***Transforming automatically into lowercase letters***

Transforming automatically into lowercase letters
Select the text to be transformed into lowercase letters, click on the icon and validate.

- ***Angle***

Select the text or characters which you wish to rotate, enter the degree of rotation, and validate.

Remark
By default, the angle of rotation is at zero since all the characters are flat on the baseline.

- ***Underlining texts***

Underlining text
Select the characters to be underlined, click on the icon **Underlining text**, and validate.
The line will appear under the baseline at a distance proportional to the height of the underlined characters.

- ***Writing from right to left***

If this button is pressed, what you write with the free text tool goes from right to left (for arabic language for instance).

- ***Variables***

Tool used to enter defined variables.
See explanations in the chapter **Variables**, section **Inserting a variable text**.

- ***Spell check (option)***

Dictionary
Select the text you wish to check and click on the icon **Dictionary**.
The spell checker should be already installed on your PC.

- ***Spacing of characters***

Proportional Spacing
Select the text or characters to be spaced, click on the icon **Proportional spacing**, and validate.

Example

Normal *at 200 %*

Remark
By default, the given spacing is at 200%, in other words, twice the original spacing.
The spacing value can be modified in the Text Attributes **box (Blue Rapido). It will become the parameter by default till the exit of the software.**

- ***Modifying character spacing***

Proportional Spacing

Select the text or characters to be spaced, click on the icon **Proportional spacing**, and validate.
The spacing by default is 200%.

Example: When the height of the character is 10mm and you enter a spacing value of 200%, the spacing obtained is 20mm.

- ***Indices and Exponents***

Indices
Select the character(s) to which you wish to add the index, click on the icon **Indices**, and validate.
By default, the distance between the text or character indexed and the baseline is equal to 50% its height.
The size of the indexed character is reduced to 50 %.

Exponents
Select the character(s) to which you wish to add the exponent, click on the icon **Exponents**, and validate.
By default, the distance between the text or character with the exponent and the baseline is equal to 50% its height.
The size of the exponent character is reduced to 50 %.

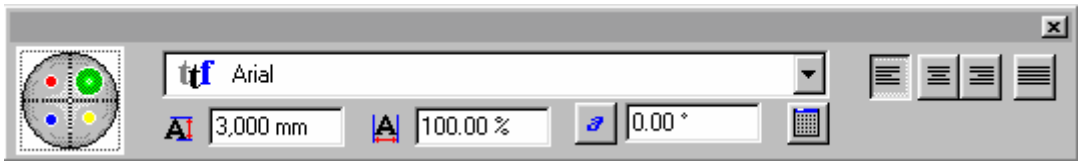
- ***Positioning the characters in relation to the baseline***

The text may be placed either above or below the baseline.
Select the text to be repositioned, click either on the icon **Text above the baseline** or on the icon **Text below the baseline**, and validate.

Text above the baseline

Text below the baseline

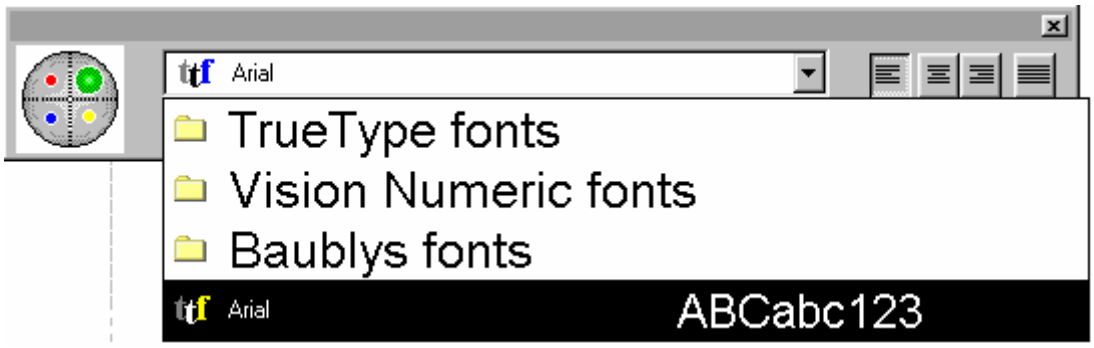
Green Rapido: Character Attributes



- ***Choosing the text font***

The library of fonts includes **TrueType** and **Vision Numeric** fonts. Other formats are available.

Click on the arrow of the scroll down menu to see a list of fonts in the library.



Use the scroll bar to select the desired font and click on it.
If you know the name of the font, you can rapidly access to it by typing the first letter of its name.

To facilitate the selection of fonts frequently used, the 5 last fonts used are displayed at the beginning of that list.

- ***Size of characters***

Height of the character in millimeters (or in the chosen unit)
With the mouse, select the text or characters to be raised or lowered, enter the desired height in mm in the corresponding field, and validate.

*Example on the first letter of the word **Rapido***

The height of R is 25 mm whereas the height of the other letters is 10 mm

Width of character in percent
With the mouse, select the text or characters to be widened or shortened, enter the desired width in percent in the corresponding field, and validate.

*Example on the first letter of the word **Rapido***

Widening of R by 500 %

- ***Modifying the height of the character***

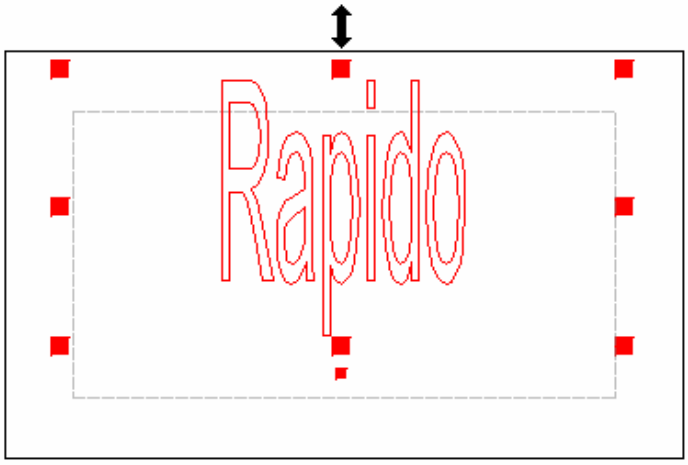
Numerically

With the mouse, select the text or characters to be raised or lowered, enter the desired height in mm in the corresponding field, and validate by clicking on the red button.

Remark
You can also modified the size of the character using the mouse.

Selection tool
Select the text or character with the selection tool and click on the handle at the top center of the text. Without releasing the button on the mouse, drag the mouse toward the top or bottom of the screen depending on whether you wish to raise or to lower the text.

Example:



Remark
You can give the accurate dimensions of the height you wish to apply by hitting the F2 key.

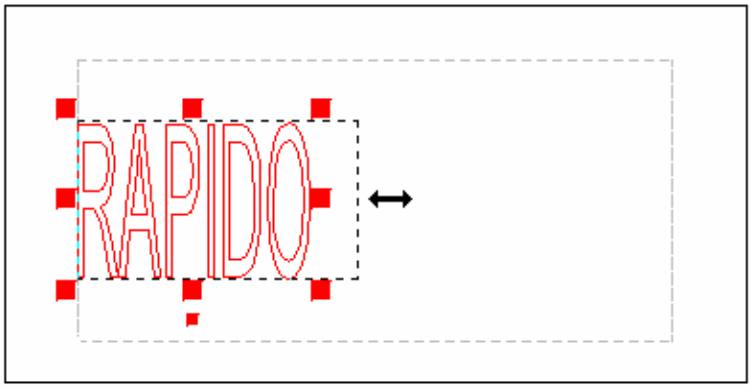
• *Modifying the width of the character*

Width of the character in percentage
With the mouse, select the text or characters to be widened or shortened, enter the width desired in the corresponding field, and validate.

Remark
You can also modified the width of the character using the mouse (see hereafter).

Selection tool
Select the text or character with the selection tool and click on the handle at the right center of the text. Without releasing the button on the mouse, drag the mouse toward the left or the right of the screen depending on whether you wish to shorten or to enlarge the text.

Example



Remark
You can give the accurate dimensions of the height you wish to apply by hitting the F2 key.

• *Slanting*

Slanting
Select the text or characters to be slanted, click on the icon **Slanting**, and validate.

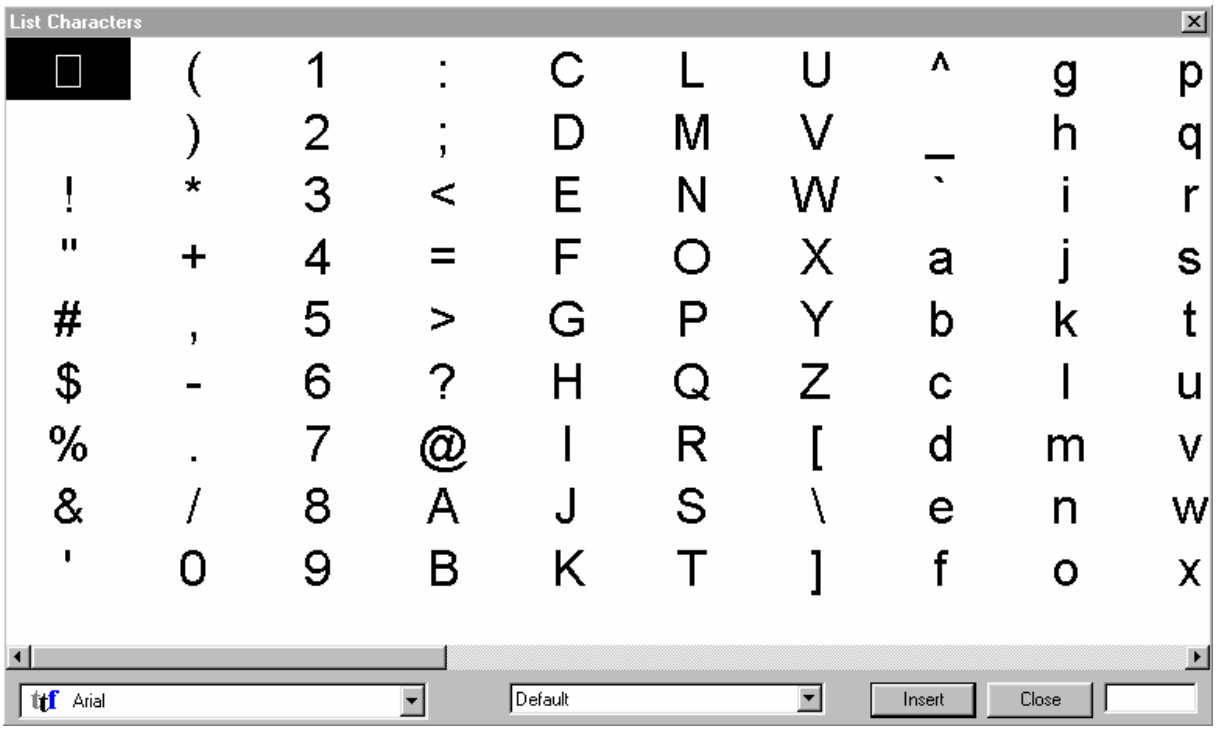
Example

at 15° *at -15°*

Remark
By default, the degree of the slant is equal to 15°.
Inserting a minus sign "-" before the slant degree inverts its direction..

- ***Previewing fonts***

Previewing fonts
You can preview the fonts before validating your choice.
Click on the icon **Previewing fonts** to open the dialog box **List of characters** and search in the library of fonts using the scroll down menu.



You can also reach this dialog box by using the icon **Font edition** . See the section **Font creation** to know how to proceed.

- ***Inserting a symbol or a special character***

This function allows you to choose a symbol or to search for a special character.

Previewing fonts
Position the cursor where you would like to insert the character or symbol, and click on the icon **Previewing fonts** of the **Green Rapido**.
In the library displayed in the dialog box **List of characters**, use the scroll-down menu to choose a font, select the desired character or symbol, validate your choice by clicking on **Insert**, and close the dialog box by selecting **Close**. The chosen character or symbol will have been inserted within the text.

- ***Aligning Text***

By default the block justification is centered. You can modify it by selecting another justification.

Align the text to the left of the starting point.

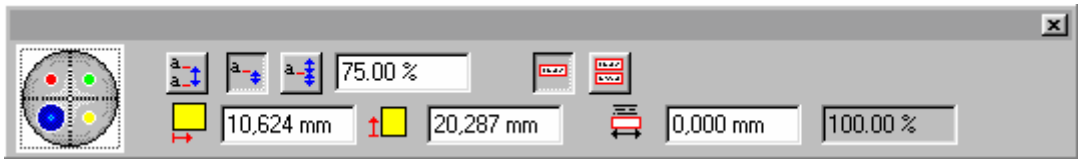
Center the text from the starting point.

Align the text to the right of the starting point.

Align the text to block with regard to the starting point (fills in the whole available space).

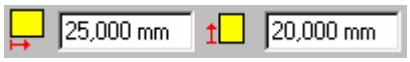
Blue Rapido: Line attributes

The **Blue Rapido** allows you to position your text, to define the length as well as line spacing and to compress either lines or paragraphs.



- ***Positioning of texts***

By default, you can position the text where you want on the material. Therefore, you just need to position the cursor on the desired position and to click on the left button of the mouse.



The coordinates of the point of beginning for the text can be numerically modified.
First select the text. Enter the new values as in the example above (**X**=25 and **Y**=20) in the corresponding fields and validate the change by hitting **Enter**.

- ***Compressing lines***

This compression can be done one line after each other or globally in such a way that the size of the characters get standardized.

Compressing lines

The benefit of this function is to compress the characters so that the text will not go over the length of the defined baseline.

Example with a 20 mm long text ("Compressed"):

Using this function allows to shorten the text "Compress". The length of this text is 20 mm and must be shortened to 15 mm.
First select the text to compress, and click on the icon **Compressing lines**.

Modify then the length of the baseline by entering the desired value in the field **Maximum length** of the **Blue Rapido**.

Validate by hitting **Enter** or by clicking with the left button of the mouse on the material. The length of the selected text is 15 mm.

• *Compressing paragraphs*

Compressing paragraphs

The benefit of this function is to compress a whole paragraph referring to the shortest line.

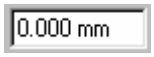
Example with a text which shortest line is 20 mm long:

The length of the shortest text must be decreased to 10 mm without altering the original ratio width/height of the paragraph.
First select the paragraph you want to compress.
Modify then the length of the baseline by entering the desired value in the field **Maximum length** of the **Blue Rapido**.

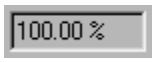
Click then on the icon **Compressing paragraphs** and validate the operation by pressing the **Enter** key or by clicking on the left button of the mouse with its cursor on the material.
The length of the shortest word has been reduced to 10 mm and the original ratio width/height of the paragraph is kept.

• *Length of text*

Distance of the maximum length of the baseline.
By default, the line stretches across the whole page between the left and right margins.



This distance corresponds to the **real size of the text before compression**, when the percentage of compression is equal to 100 %.
Therefore, since the length of the text cannot be superior to the length of the baseline, the characters will be squeezed and the percentage of compression will become inferior to 100 % when the length of the text is increased.



Percentage of compression of the text on the baseline.
This value is at 100% before compression and decreases when the length of the text is larger than the distance of the baseline.

• *Parametrical line spacing*

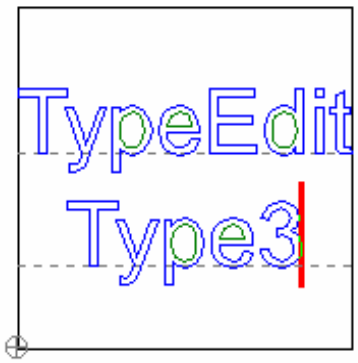
Parametrical line spacing
Use this function to set the space between lines.
The interval expressed in the default value defines the distance between the two baselines without taking the height of the characters into account.

• *Proportional line spacing*

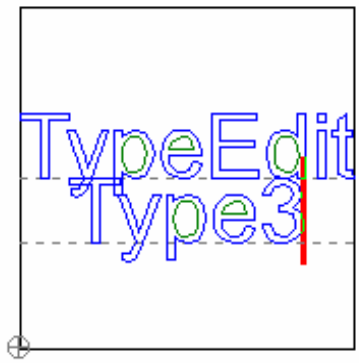
The spacing between the lines is expressed in percent and depends on the text font.

This line spacing parameter takes into account the height of the superior downstroke i.e. the characters above the base line. By default, it is 75% of the height of the text.

Example



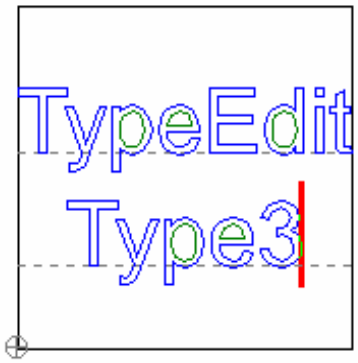
at 75%



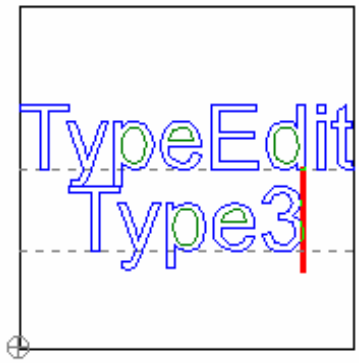
at 0%

The typographic line spacing takes into account the whole height of the characters, superior and inferior downstrokes included.
By default, this parameter is 100% of the height of the text.

Example



at 100%



at 0%

Red Rapido: Styles

To facilitate access to often used texts or characters you can save the styles: font, dimension of the character, slant, etc. To save styles, use the **Red Rapido**.

- ***Creating a style***

Define the attributes parameters for characters or paragraphs in such a way that the text appears as desired: font, slant, position, etc.

Click on the **Red Rapido** and enter the name of the style in the field of the scroll down menu.

Click on **Save** to save this style.
The style includes all the defined parameters at the time when saved.

- ***Finding a style***

Choose a style from the scroll down menu, validate your choice, and type your text.
If the text is already typed, select it, choose the desired style from the scroll down menu, and validate your choice.

Remark
The last used style is always the one that remains active.

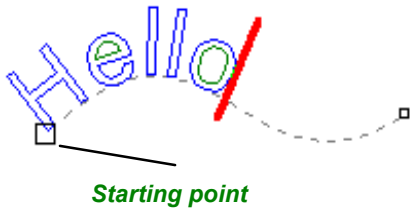
Text on a Curve

Text on a curve
You can bracket a text on any shape: a straight line, a curve, a rectangle, an ellipse, etc.

Trace your baseline with the help of the **Sketching tool** .
Then click on the icon **Text on a curve**. The **Rapido** is displayed on the screen allowing you to define all the parameters (styles, positioning, etc.). Create your text.

Orientation of the text

The beginning point of the tracing defines the starting direction of the text.
The orientation of the text (from left to right or from right to left) depends of the direction of the tracing chosen when drawing the baseline.
Hence, a tracing, no matter how created, if it is drawn from left to right, the text will appear from left to right.

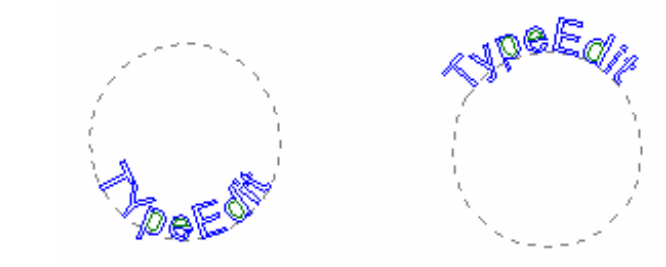


On the other hand, a tracing created from right to left will make the text be positioned in the opposite direction from above, in other words, from right to left.

Click on the icon **Reverse contour** to invert the direction of the contour.

Remark

In the case of a closed contour, it is the direction of the contour (clockwise or anti-clockwise) which determines the orientation of the text.



Clockwise (green) *Anti-clockwise (blue)*

Text inside a rectangle

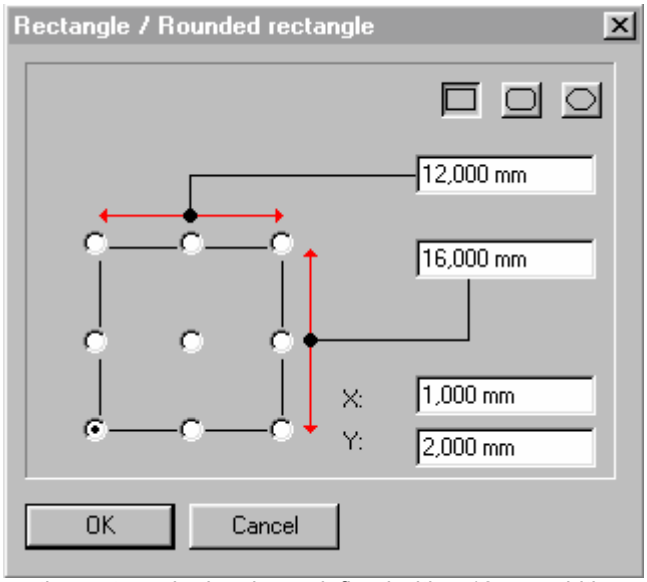
Text inside a rectangle

The interest of this function is to define a rectangular area from which the text cannot go beyond.
As a consequence, the text contained in this area will be automatically centered and compressed.

Remark

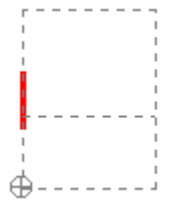
You can either apply this function to an existing text (you need to select the text) or define the rectangular area before creating the text.

Click on the icon **Text inside a rectangle**.
The dialog box **Rectangle/Rounded rectangle** is displayed.



In the example shown above, the rectangular has been defined with a 12 mm width and a 16 mm height.
This area is positioned referring to the left inferior corner point which is activated in the dialog box of the example.
The coordinates of this left inferior corner point are **X**=1 mm and **Y**=2 mm.
Click on **OK** to validate.

The rectangular area is displayed on the screen. The cursor is automatically centered vertically.



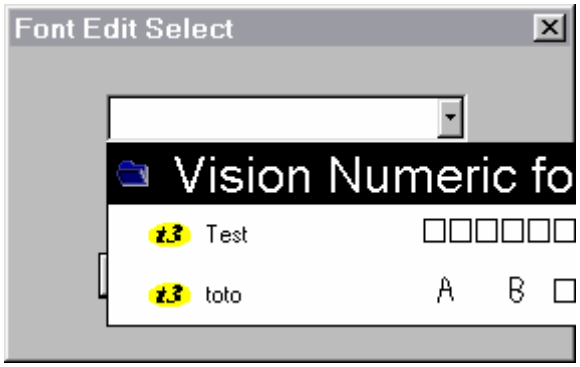
Create now your text: this one is automatically compressed to pass into the defined area.

Creation / modification of a font

The software includes a font editor that allows you to create or modify one by one the characters of an existing font.

Modification of a font

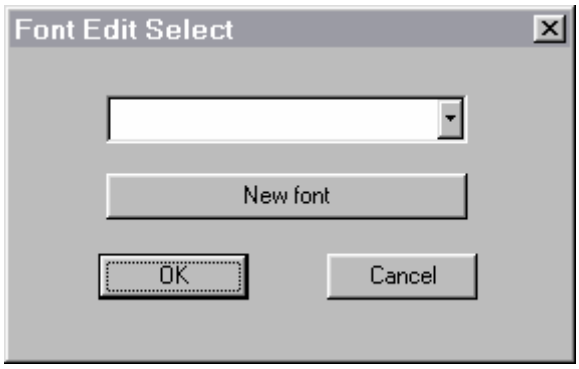
Click on the **Text mode** icon to display the toolbar where you can find the **Font edition** icon . The dialog box **Font edit select** is displayed on the screen:



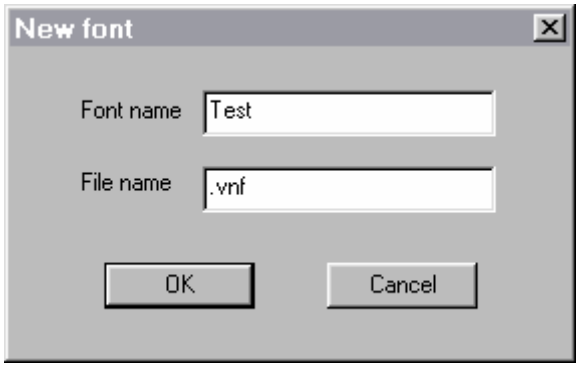
Choose in the scroll down menu the font you want to modify and click on **OK.** Then proceed the same way as for the creation of a new font (see further).

Creation of a new font

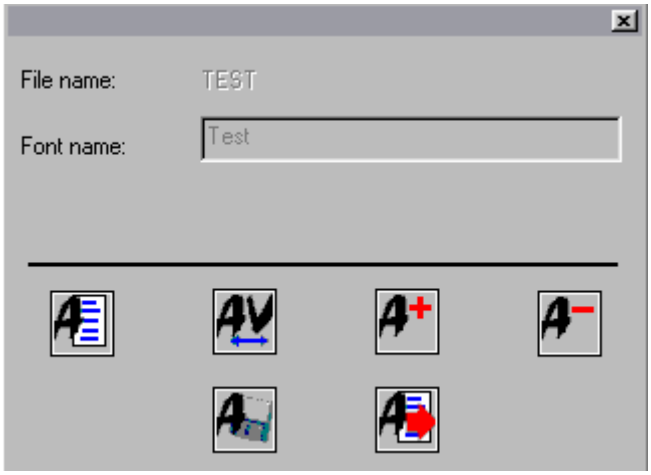
Click on the **Text mode** icon to display the toolbar where you can find the **Font edition** icon . The dialog box **Font edit select** is displayed on the screen:



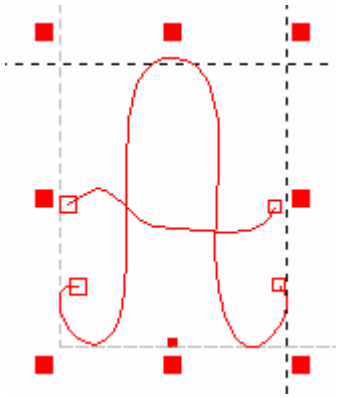
Click on **New font**. The following dialog box allows you to name your font:



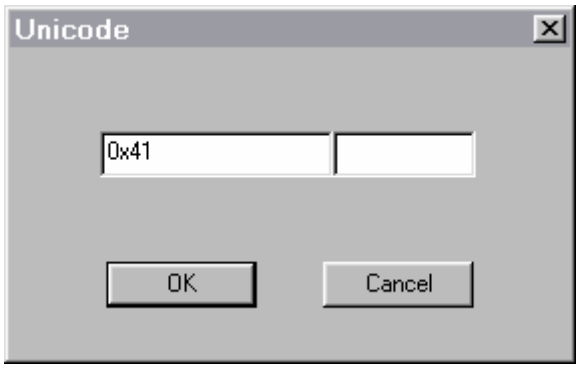
Give a name to the font and click on **OK**. Your font appears in the library. Click again on **OK**. The next dialog box allowing you to create your new font and the kerning between characters is displayed on the screen.



First of all, draw a guide line that allows to define precisely the height and width of the letters, as well as the spacing before and after the letters. Then, draw you letter with the modeling tools or all other necessary tools.



Select the whole letter and push the button. Whatever is in Layer 1 of your vnd file will be included in the character, so you have to position the lower left corner of your character at the origin of your layer. Also note that your character should be 100 mm high in order to keep a 1:1 scale when choosing the size of your font. The following dialog box appears on the screen:



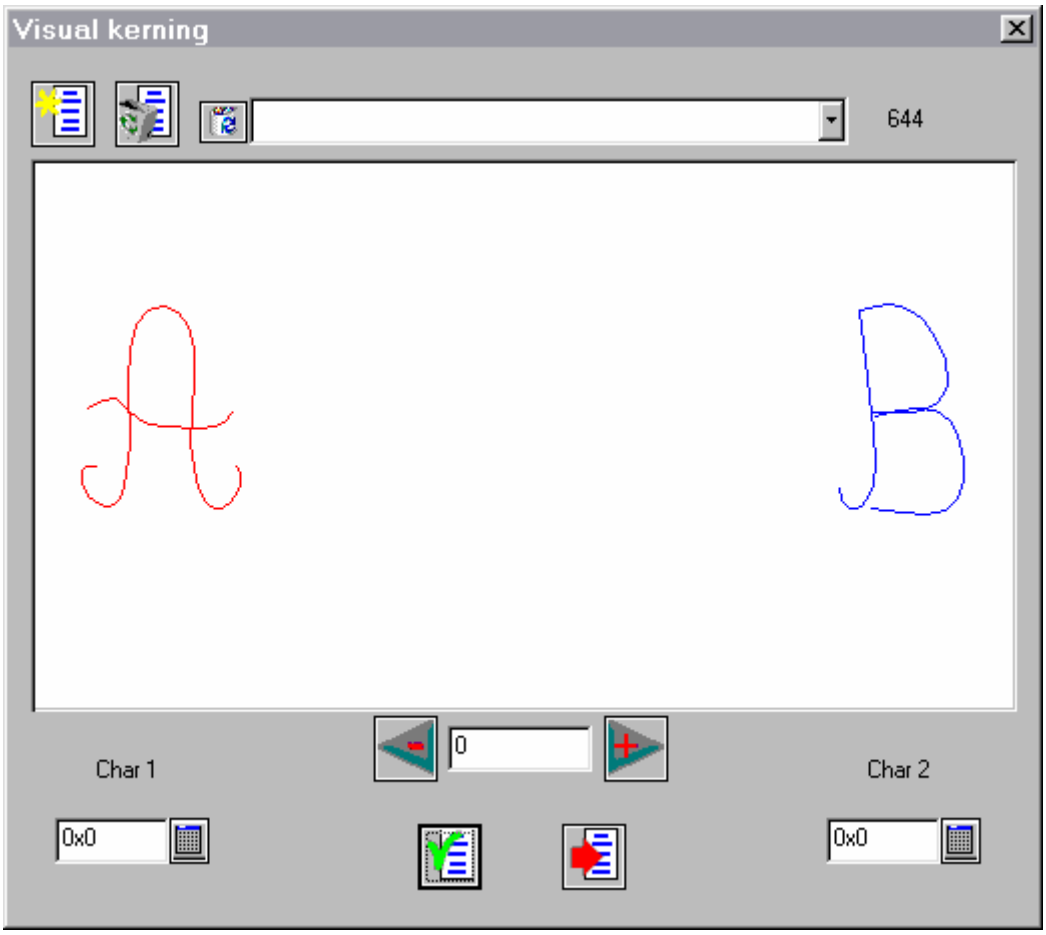
In the right box, enter the letter you've just created (be careful with the small and capital letter!). **Nothing appears on the screen**. However, the letter code appears in the left box and allows you to identify the character you saved. Confirm with **OK**. If you wish to save your character, push the button.
Do the same for the other letters of the alphabet. Don't forget to save your work after having created a letter.

To erase a letter, push the button. Select the letter you want to erase and close the dialog box. Push the button to save your modification.

The icon allows you to visualize the list of available characters

The icon allows you to close the font editor.

The icon allows you to define the kerning between letters.



Char 1 and **Char 2** allows you to define the letters between which you want to define the kerning. Click on the icon **Char 1** to choose the first character in the list. Proceed the same way to choose the second one. Both of the selected characters are displayed on the screen. You can enter the value of the kerning in the middle box. To refine your evaluation, use the buttons . Confirm your choice by clicking on .

With the scroll-down menu, you can list the kerned letters.

The icon allows you to choose the kerning recommended.

The icon allows you to erase all kernings.

CREATING OBJECTS

Drawing geometrical shapes

This section deals with creating an object by free hand and with the help of tracing shapes (square, oval, circle, starburst...).

Shape tool
This icon opens the toolbox with the different geometrical shapes available.

Remark

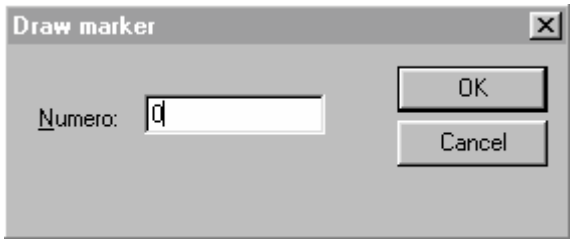
The drawing of an object can always be made numerically. The numerical functions are continually active when you create, move, or resize an object with the F2 key.
You can put any selected tool back down by hitting the space bar.

To draw these geometrical shapes, select the tool corresponding to the shape you want to draw and place the cursor, which represents one of the tops of the shape, on the active area.

Marker

Marker
To create a point, click on the icon **Marker**. Then place your marker on the material by simply clicking on the left button of the mouse.

Ten different kinds of marker are available. They are numerically ordered from 0 to 10. By default, the marker N° 0 is displayed.
To modify the graphical representation of the marker, press the **Ctrl** key at the same time you click on the icon **Marker**. The dialog box **Draw marker** is displayed.



Enter the ordered number corresponding to the graphical representation of the marker you want to use. Below is the list of the ten markers available:

- Marker N° 0
- Marker N° 1
- Marker N° 2
- Marker N° 3
- Marker N° 4
- Marker N° 5

- Marker N° 6
- Marker N° 7
- Marker N° 8
- Marker N° 9

Validate with **OK** and position the marker on the material.

With **F2**: gives the accurate coordinates of this point.

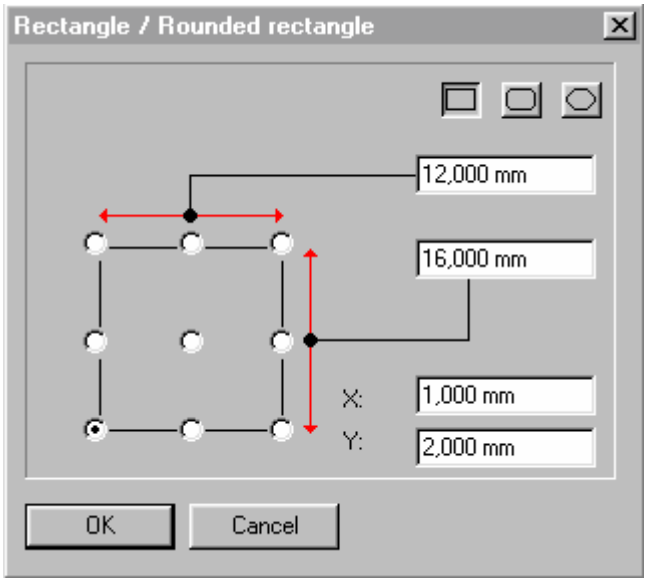
Rectangle

Rectangle

Click on the icon **Rectangle**. Position the cursor on the material, and while holding down the left mouse button, drag the mouse until your rectangle is set. Then release the mouse button.

Trace while pressing the **Ctrl** key = Square
Trace while pressing the **Ctrl + Shift** keys = Centered square

You can also draw your rectangle numerically. Therefore, click on the icon **Rectangle** and then press the **F2** key. The dialog box **Rectangle/Rounded rectangle** is displayed on the screen.



First select the point which coordinates **X** and **Y** will be defined in the corresponding fields of the dialog box. Then give the width (12 mm in the above example) and the height (16 mm in the same example).
Click on **OK** to validate.

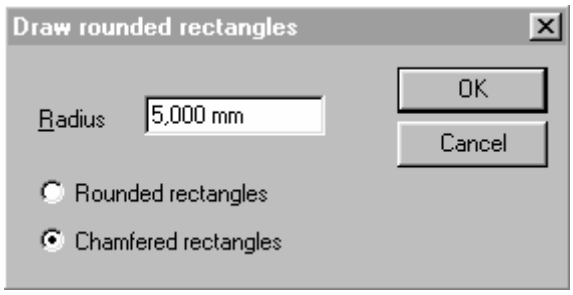
Remark

Clicking on or allows to create a rectangle either with rounded corners or with chamfered corners (see next section).

Rectangle with rounded or chamfered corners

To trace a rectangle with rounded or chamfered corners, click on the icon **Rectangle with rounded or chamfered corners**.

In the dialog box **Draw rounded rectangles** that appears, type the radius you want and indicate whether you require rounded or chamfered corners as shown hereafter.

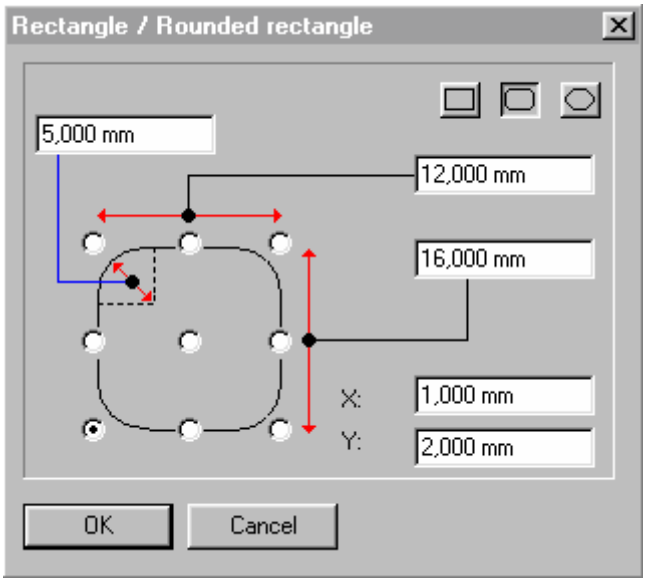


Follow the same procedure as for the tracing of a rectangle using either the mouse or **F2**.

If you enter negative values, arcs will be inside the rectangle:

Trace while pressing the **Ctrl** key = Square
Trace while pressing the **Ctrl + Shift** keys = Centered square

You can also draw a rectangle with rounded angles by clicking first on the icon **Rectangle**, then on **F2** and finally on the icon within the dialog box **Rectangle/Rounded rectangle** that is automatically modified. Compared to the original dialog box, there is an added field to fill in. This value represents the radius of the rounded corner.



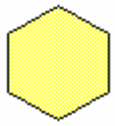
Follow then the same steps as for the creation of a rectangle as described before.

Click on the icon in the dialog box **Rectangle/Rounded rectangle** to draw a rectangle with chamfered corners numerically and follow the same steps as for the creation of a rectangle with rounded corners.

Polygon

Polygon

To trace a polygon, click on the icon **Polygon** and enter the number of sides desired in the dialog box **Draw polygons** that is displayed on the screen.

Example: 

Remark

You need to enter a value greater than or equal to [3].

Trace while pressing the **Shift** key = 2 opposite points of the included circle

Star

Starburst

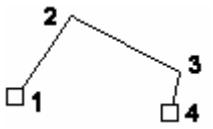
To trace a star, click on the icon **Starburst**.
In the dialog box **Draw stars**, enter the number of vertices desired.

Draw a circle corresponding to the external size of the star, then click on the left button of mouse.
Drag the cursor towards the inside of the circle and click again on the left button of the mouse to finish.
The closer you are to the center, the thinner the vertices of the star will be.

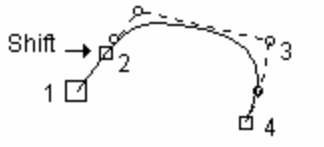
Sketching tool

Sketching tool

To draw lines or curves by free hand as well as shapes associating lines and curves, click on this icon **Sketching tool**.
Click once on any point and drag. Each time you release the mouse button, you create a line.
Double-click to stop.



Trace while pressing the **Ctrl** key
= angular point (segment)



Trace while pressing the **Shift** key
= tangent point **after a segment**

You can use the **F2** key to draw a segment with exact coordinates.
Select the **Sketching tool** and click on **F2**. Enter the X1, Y1, and Z1 coordinates for point 1.
Then click on **F2** to enter the X2, Y2, and Z2 coordinates for point 2.
Finally click again on **F2** and enter the same coordinates X2, Y2 and Z2 of the point 2 to end the operation.

Double line

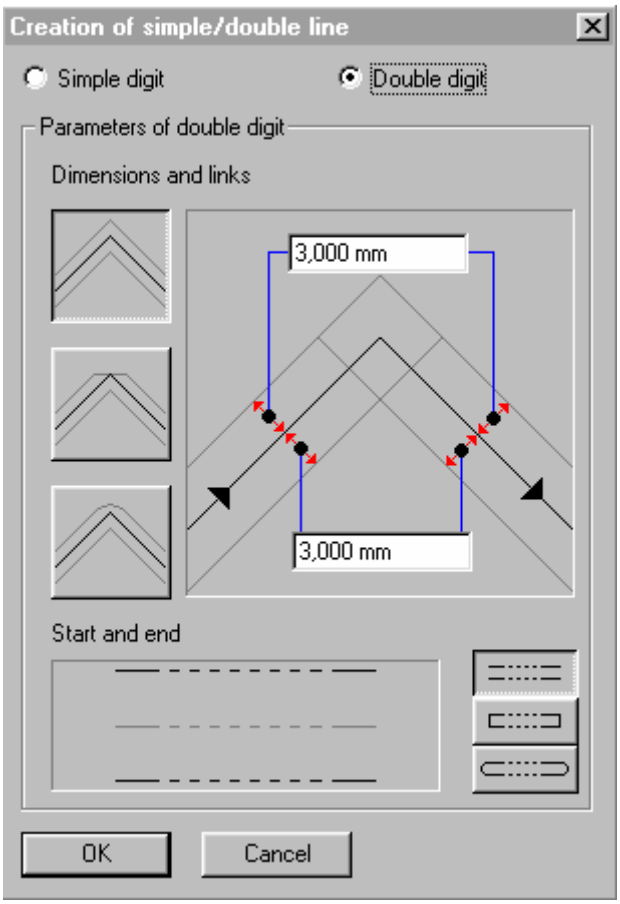
Double line

The interest of this function is to trace double lines as well as single line segments (line segments can also be drawn by pressing the **Ctrl** key while tracing with the **Sketching tool**).

Drawing a double line

To draw a double line, click on the icon **Double line**.
You can set the parameters of this tool by pressing the **Ctrl** key while clicking on the icon **Double line**.

The dialog box **Creation of simple/double line** appears on the screen.



First choose the kind of link and its dimensions. Therefore, click on one of the three icons:

Creation of sharpened angles

Creation of chamfered angles

Creation of rounded angles

Define then the spacing between each line (of the double line) and the median axis by entering the numerical values in the corresponding fields.

Remark

In case you create an open contour, you can also define the double line's aspect at its starting point and at its arrival point.

To define the double line's aspect at its start and at its end, click on one of the three icons:

Open double line

Closed double line with sharpened angles

Closed double line with rounded angles

After having selected the desired options, click on **OK** to validate.
To start drawing the double line, click once on the material and drag using your mouse.

Click on the mouse each time you want to create an angle.

If you want to trace parallel to the axis **X** or **Y**, keep pressed the **Ctrl** key while dragging the cursor.

Remark

You can numerically define the position and the value of each angle by hitting the F2 key and entering numerical values.

To validate an open contour, double-click as soon as you want to end its tracing. The contour is displayed in red.

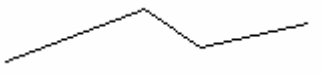
To close a contour, drag the cursor until the starting point. When the contour becomes blue or green, it is closed. Double-click on the mouse's left button to validate your contour. The contour appears in red.

Drawing a simple line

To trace a simple line, select the option **Simple digit** of the dialog box **Creation of simple/double line**.

To display this dialog box, keep pressed the **Ctrl** key while you click on the icon **Double line**.

To start drawing the simple line, click once on the material and drag using your mouse. Click on the mouse each time you want to create an angle.



If you want to trace parallel to the axis **X** or **Y**, keep pressed the **Ctrl** key while draging the cursor.

Remark

You can numerically define the position and the value of each angle by hitting the F2 key and entering numerical values.

To validate an open contour, double-click as soon as you want to end its tracing. The contour is displayed in red.

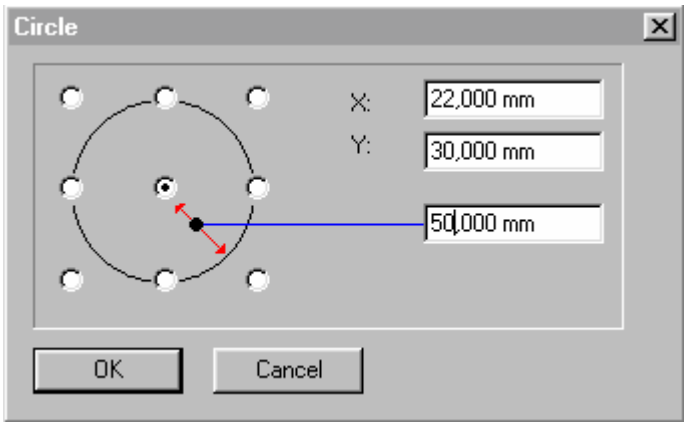
To close a contour, drag the cursor until the starting point. When the contour becomes blue or green, it is closed. Double-click on the mouse's left button to validate your contour. The contour appears in red.

Circle and ellipse

Circle

To trace a circle, click on the icon **Circle**, then on the active area.
Drag the mouse maintaining the left button pushed until the object obtains the desired dimensions. Then release the button to finish.

With **F2**: enter the accurate dimensions of the circle, its radius (50 mm in the example hereafter) and the coordinates **X** and **Y** of a point you can select among the nine points of the dialog box **Circle**.



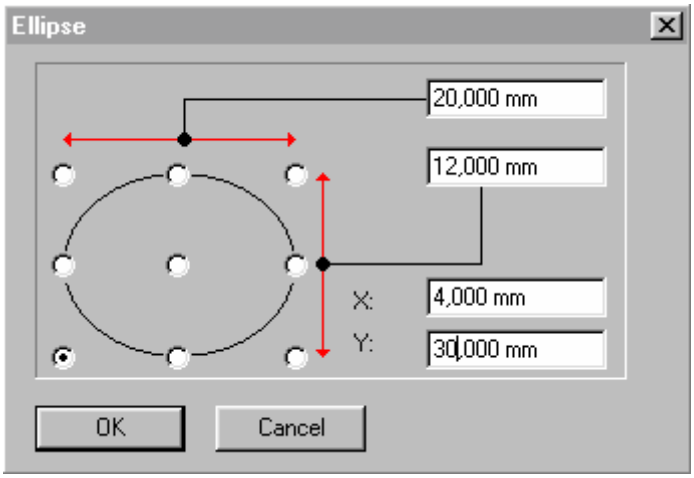
Ellipse

To trace an ellipse, click on the icon **Ellipse**, then on the active area.
Drag the mouse maintaining the left button pushed until the object obtains the desired dimensions. Then release the button to finish.

Trace while pressing the **Ctrl** key	= circle
Trace while pressing the **Ctrl + Shift** keys	= ellipse centered in its circle
Trace while pressing the **Shift** key	= centered ellipse

With **F2**: enter the accurate dimensions of the ellipse in the dialog box **Ellipse**:

- The width (20 mm in the example shown hereafter)
- The height (12 mm in the example shown above)
- The coordinates **X** and **Y** of the selected reference point.



Circle (3 points)

To trace a circle defined by 3 points, click on the icon **Circle (3 points)**.
Indicate the three points on which you want to draw the circle. Then draw the figure by clicking on the first two points to begin the drawing and by validating it on the third point.

Remark

If you click once on F2, you can directly enter the coordinates of the first point of the arc of the circle.

Repeat the operation a second time, then a third time to respectively enter the coordinates of the second and third points of the arc.

Arc of circle

Arc

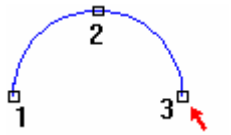
To trace an arc, click on the icon **Arc**.
Draw the circle in which the arc will be formed.
Once the circle is drawn, release the button at a certain point which will be the starting point for the arc.
Drag the cursor until the arc has the desired dimension, and then validate.

Remark

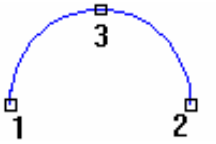
While pressing the Ctrl **key, the tracing will be in the clockwise direction.**

Arc (3 points)

To trace an arc defined by 3 points, click on the icon **Arc (3 points)**.
Indicate the three points on which you want to draw the arc.



Then draw the figure by clicking on the first two points to begin the drawing and by validating it on the third point.
To define the curvature of the arc at the end, press **Shift** before validating the third point.



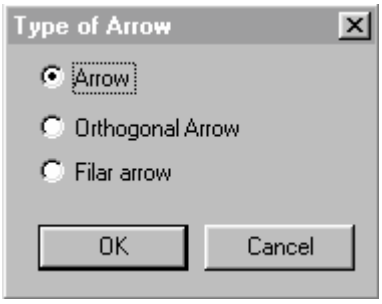
Remark

If you click once on F2**, you can directly enter the coordinates of the first point of the arc of the circle. Repeat the operation a second time, then a third time to respectively enter the coordinates of the second and third points of the arc.**

Arrow

Arrow

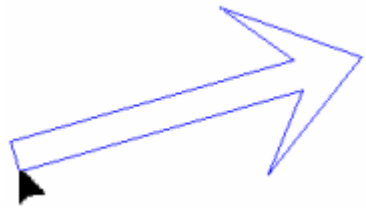
To draw an arrow, click on the icon **Arrow**.
The dialog box **Type of arrow** is displayed on the screen.



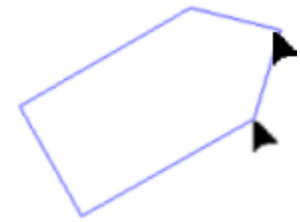
Three kinds of arrows are available. Select one of them and click on **OK** to validate.

Arrow

Select **Arrow** in the dialog box **Type of arrow** and click on **OK** to validate.
Create the arrow's base by clicking on the material (the base is pointed out by the cursor on the picture hereafter).
Drag the cursor to define the direction of the arrow, and click a second time when the arrow has got the desired length (from the base to the head).



Drag now the cursor to define the global width of the arrow as well as the head's angle. Click a third time when the arrow has got the desired shape and dimensions.

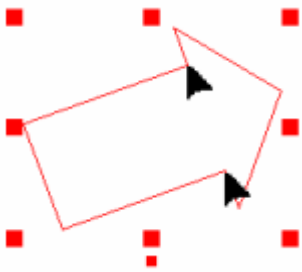


Drag the cursor to define at the same time the width of the arrow's tail and the angle of the lateral tips. Click a last time to validate the arrow you just created. The arrow gets displayed in red.

Orthogonal arrow

This kind of arrow features right angles at the junction of the arrow tail and its lateral tips.

Example



To draw an orthogonal arrow, follow the same steps as for the creation of a "classical" arrow, after having selected **Orthogonal arrow** in the dialog box **Type of arrow**.
The created orthogonal arrow gets displayed in red.

Remark

If you are creating an orthogonal arrow, the last drag of the cursor before validation will not allow to define the angles.

Filar arrow

This kind of arrow is represented by three segments.

Example

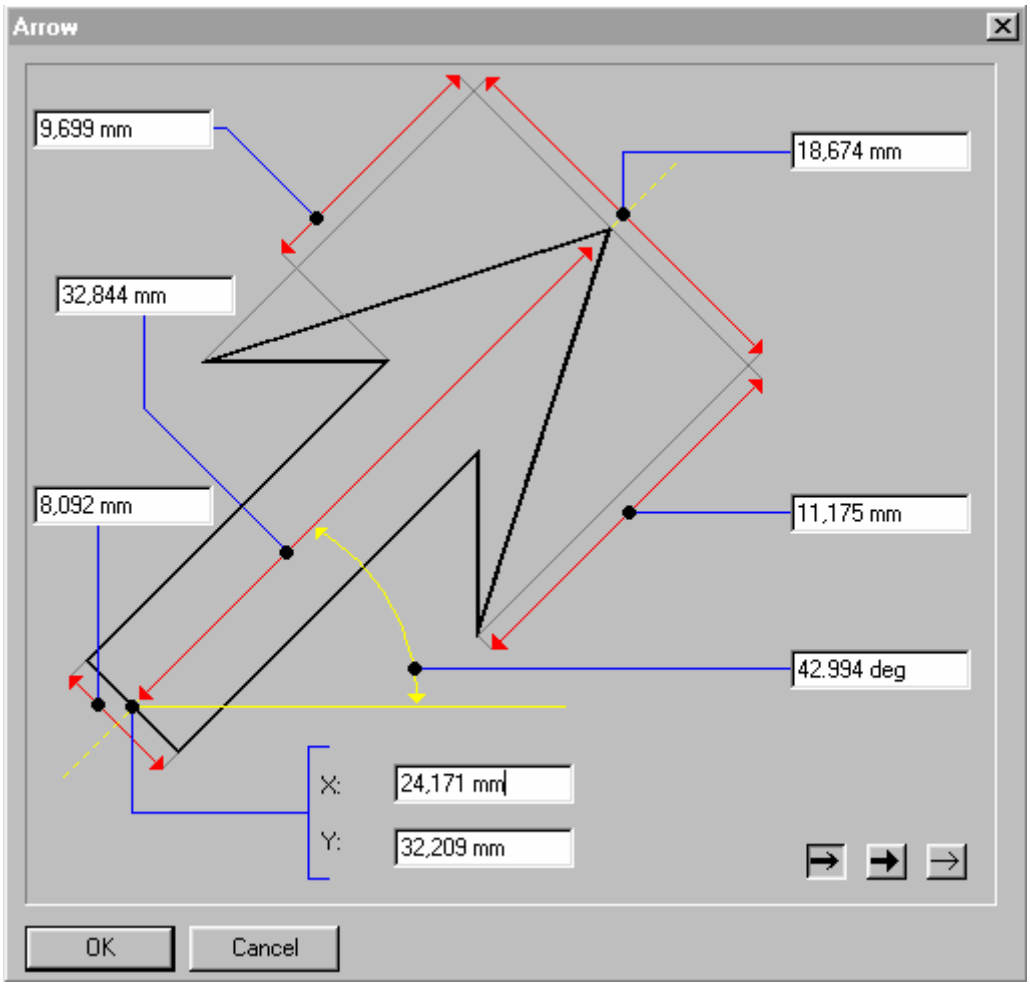
To draw a filar arrow, follow the same steps as for the creation of a "classical" arrow, after having selected **Filar arrow** in the dialog box **Type of arrow**.

The created filar arrow is displayed in red.

Drawing an arrow numerically

To draw arrows numerically, click on the icon **Arrow**.
Select the kind of arrow you want to create in the dialog box **Type of arrow** and click on **OK** to validate.

Click then on the material to begin drawing the arrow and define its direction.
Press then the **F2** key.
The dialog box **Arrow** is displayed on the screen.



Enter the desired numerical values in the dialog box. When finished, click on **OK** to validate.

Remark

The dialog box Arrow **immediately takes into account the kind of arrow you selected. Nevertheless, this dialog box also features icons allowing you to modify the kind of arrow you want to create simply by clicking on one of them.**

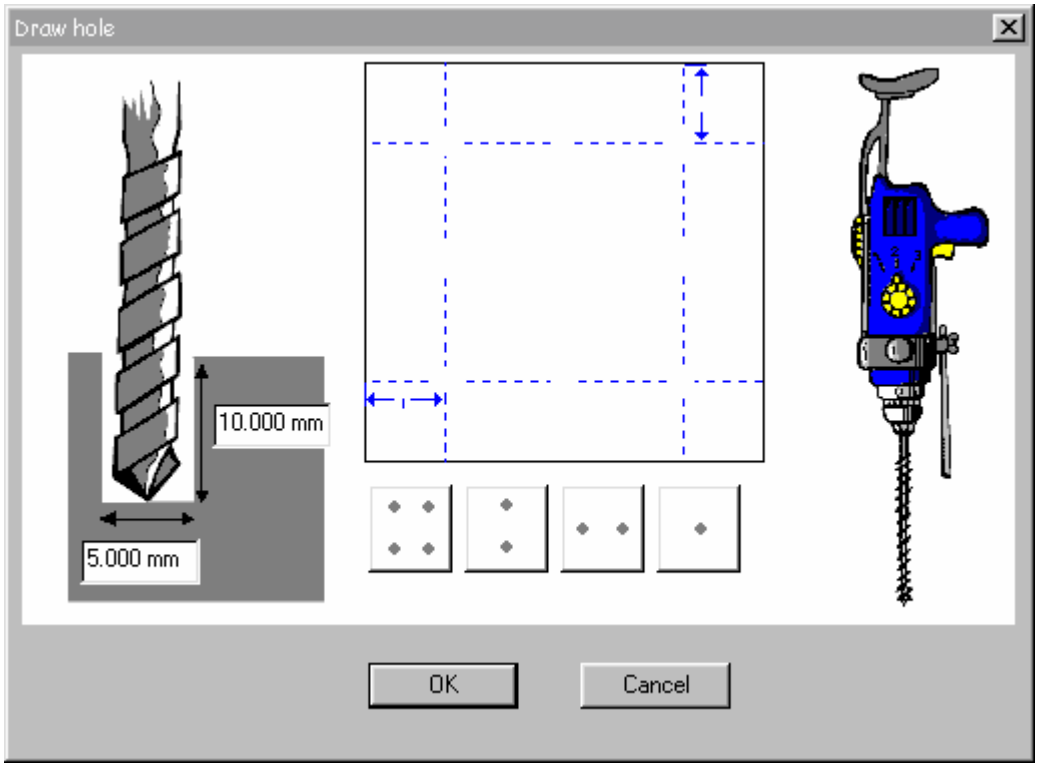
DRAW HOLES

Inserting draw holes

Draw hole

This function allows to geometrically position one to nine draw holes on the active surface. So can you take into account the future positioning of the holes in relation to your composition.

Click on the **Draw hole** icon. The dialog box **Draw hole** is displayed on the screen.



First enter the dimensions (depth and width) in the two fields located in the lower left corner of the dialog box. Select then the positioning of the draw hole(s).

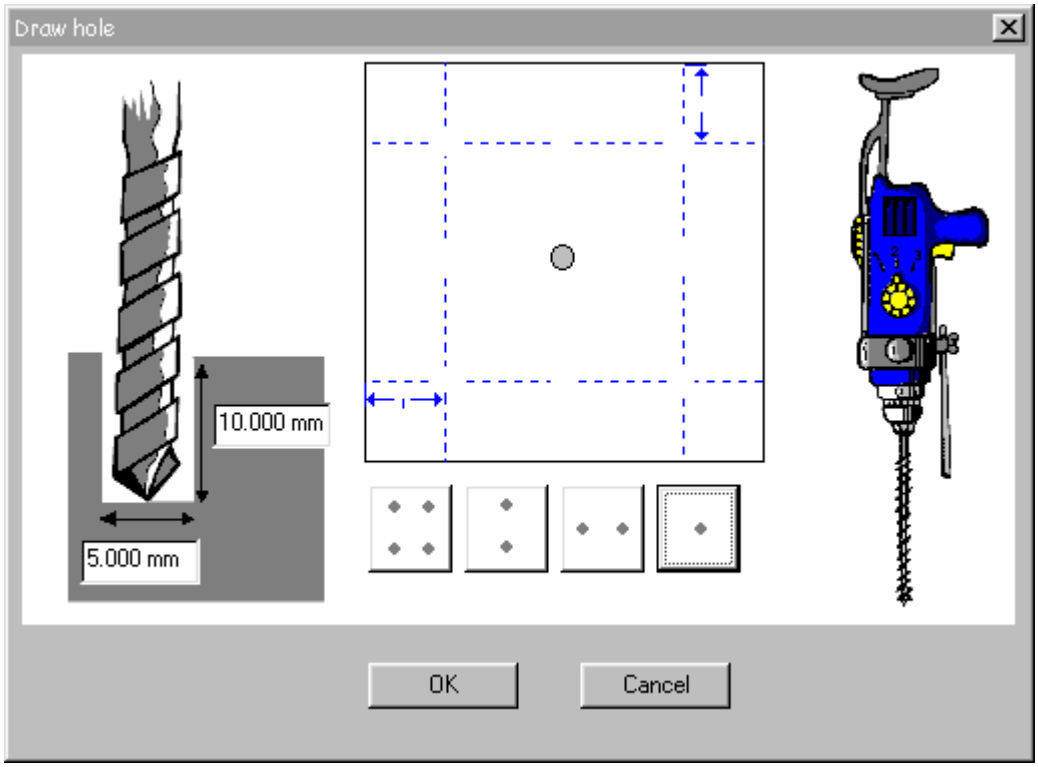
Remark

The Draw hole **function includes nine predefined positions that you may combine or no.**

Example 1

To center a draw hole on the active surface, click on the *icon.*

You can immediately see the result in the dialog box.



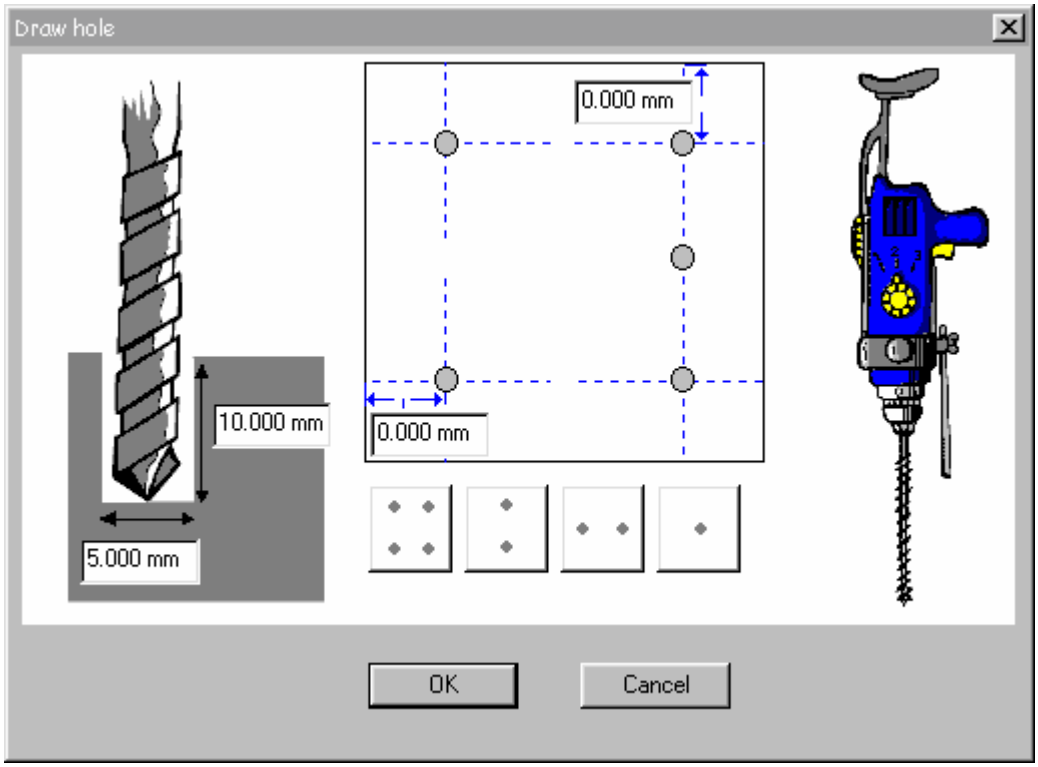
The draw hole is automatically centered.

You can also combine several positioning patterns as well as suppress some holes simply by clicking on the ones that must be deleted.

Example 2

To position five draw holes, first click on , *then on* .
Click then on the point you want to delete.

The dialog box immediately reports your selection.



Remark

You can also position the points numerically in relation to the horizontal and vertical rulers. Therefore enter the desired numerical values. This function cannot be enabled in case of a single centered point.

Once the draw holes are defined, click on **OK** to validate. The selected points are represented by ⊕ on the active surface.

Free mode

The **Draw hole** function can be used in free mode. Click on the **Draw hole** icon. The **Draw hole** dialog box is displayed on the screen.

Validate by clicking on **OK** *without having selected a positioning pattern*.

This allows you to position draw holes by clicking on the active surface (one click per draw hole to position).
Do the same numerically by hitting the **F2** key and entering the coordinates in the dialog box displayed on the screen.

Once the draw holes are defined, click on **OK** to validate. The selected points are represented by ⊕ on the active surface.

Rulers and dials: parameters

To create a vernier, you first have to create the original drawing on which the graduation will be braced. This drawing (support drawing) can be a line, a circle, or a complex shape.
The principles of graduation remain the same whatever the original shape may be.

The latest evolutions of the software allow you now to directly create the support drawing during the use of this function or to work with an existing drawing.

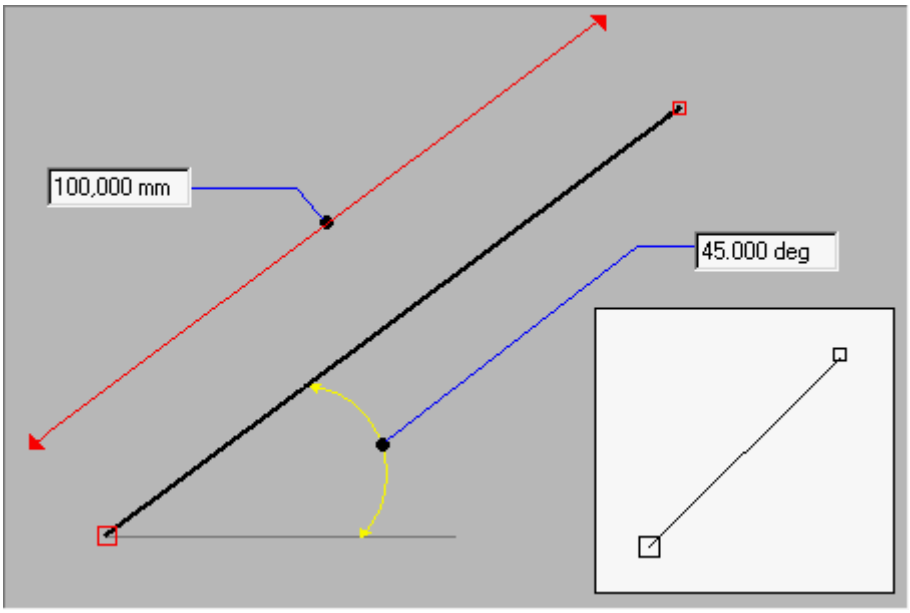
1. The support drawing doesn't exist

Click on the **Dial** icon . The dialog box **Basic definition** appears on the screen. It will help you in creating the support drawing.

Geometry

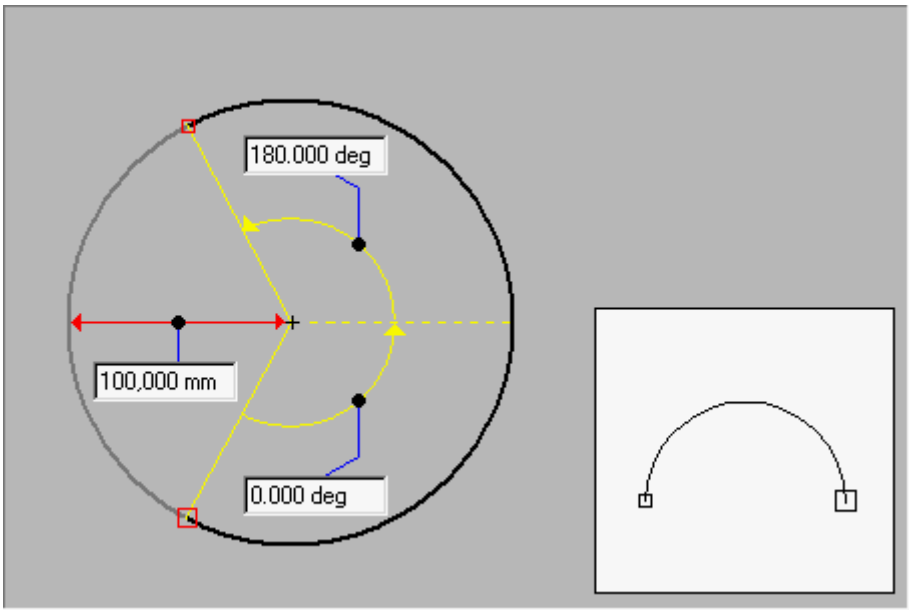
- Select the option **Linear shape** to realize a ruler. Define the position of the support drawing's starting point in the numerical fields **Origin X** and **Origin Y**. Define also the length of the ruler (100 mm in the example shown below) as well as its skew angle (45 degrees in the example shown below). The biggest square represents the starting point of the drawing.

Example



- Select the option **Circular shape** to make a dial. Define the position of the drawing support's (circle or arc of circle) center in the numerical fields **Centre X** and **Centre Y**. Define also the radius (100 mm in the example shown below), the positioning and the length of the drawing through its starting and ending angle (180° and 0° in the example shown below). Define this angle in the top field to obtain an arc towards the top (like in the example) or in the lower field to obtain an arc towards the bottom. Both angles are defined in relation with the horizontal axis. The biggest square represents the starting point of the drawing.

Example



The **Offset** checkbox allows to add a parallel line to the support curve, with an offset you specify.

Remark

Whatever the chosen shape, the preview window situated in the right bottom corner of the dialog box Basic definition **gives you an instantaneous view of the created support drawing before validating. The big square represents the starting point. Click on this zone to immediately refresh the view.**

Once all parameters are set, click on **Next** to complete the next steps.

Remark

Without having defined the graduations until now, you may be unsastified with the final result. In that case, you don't need to redo all the steps one by one. Simply double-click on your composition. The dialog box will be displayed on the screen again: you can modify the parameters.

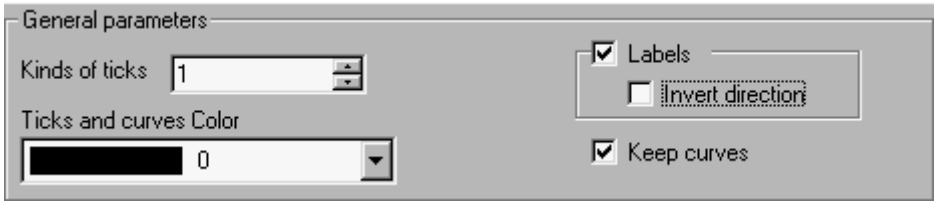
2. The support drawing already exists

Select the contour which you would like to be graduated and click on the icon **Dial** .
The dialog box **Wizard** is displayed on the screen.

General parameters

Once you've created the support drawing and clicked on **Next**, or selected an existing shape, click on the **Dial** icon . The dialog box **Wizard** gets displayed on the screen. It will help you to define the general parameters of your dial or ruler.

*Extract from the **Wizard** dialog box*



Kinds of ticks
Define how many kinds of ticks will be used.

Labels
Activate this function if you want the labels to appear on your ruler or your dial.

Keep curves
Activate this function to keep the constructed original shape.

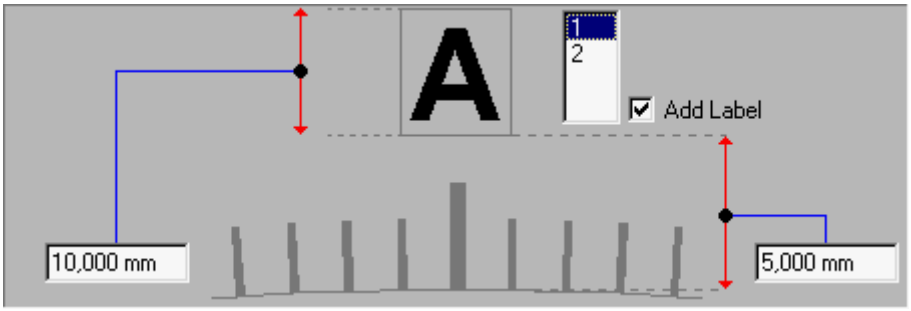
Ticks and curve color
Use the scroll-down menu to select the display color of curves and graduations.
As soon as you have defined these parameters, click on **Next** to validate. The dialog box **Labels** is displayed on the screen.

Defining labels

Once you defined the **General parameters**, and you validated by clicking on **Next**, the dialog box **Labels** is displayed on the screen. Use the **Labels** dialog box to parameterize the position, the numbering and the size of the labels.

Height, position and choice of labels

*Extract from the **Labels** dialog box*



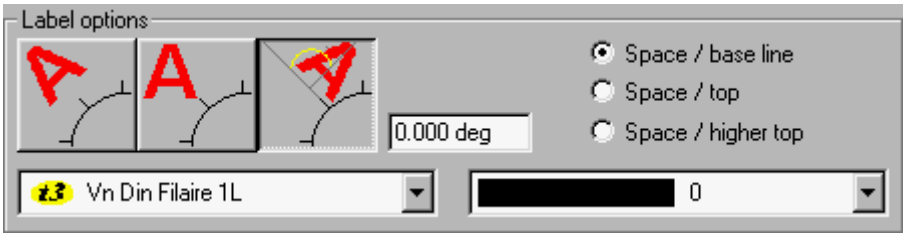
Activate the option **Add Labels** if you want to see labels on your ruler or dial. A numerical field is located next to the option **Add Labels**. In the example shown above, two kinds of ticks have been defined during the previous step (Dialog box **Wizard** → option **Kind of ticks**).
Select the kind(s) of graduations (**1** in our example) that must feature labels. Click therefore on each kind while pressing the **Ctrl** key.

Remark
Whatever the number of different ticks that have been defined, you can only define one kind of labels.

Define then the height of labels in the left field of the box upper part (10 mm in our example). Define also in the right field of the box upper part the distance between the base of the label and the base or the top of the graduation (See a little bit further the section **Referencing the label's position**…).

Position of labels

*Extract from the **Labels** dialog box*

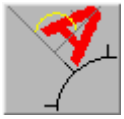


In both fields of the upper part of the dialog box, enter the values corresponding to the top the height of the label and the distance that separates it from the original shape. In the example shown above, labels have an height of 10 mm and the distance separating them from the original shape is 5 mm.

Label's slanting

Activate this function to align labels vertically.

Activate this function to align labels in relation to the original shape.


Activate this function to align labels in relation to a slanting angle to be defined in the field next to this icon.

Referencing the label's position
Select the option **Space / base line** to define the distance between the base of label and the base line (support drawing).
Select the option **Space / top** to define the distance between the base of label and the top of the graduation.
Select the option **Space / higher top** to define the distance between the base of label and the higher graduation (in case you've got numerous kinds of graduations).

Selecting the label's font
Use the scroll-down menu to select the font to be used to generate the labels.

Selecting the label's display font
Use the scroll-down menu to select the label's color to display on the screen (black in our example).

Label generation

Count for each graduation
Activate this option to make the numbering take each graduation into account. A significant example of use of this function is the making of speedometers. Currently, most of the speedometers only feature the tenth of kilometers on the best graduations.

Use numbers or **Use variables**
The values on the graduations are either numbers that you define with the three following parameters, either the values of a variable set with the tool. This last choice allows for more flexibility, above all the use of letters and various non-numeric characters.

Start
Enter in this field the starting value of the numbering.

Step (increment)
Enter in this field the value defining the step of the numbering.

Nb of decimals
Define there the number of decimal places if the start value or the incrementation step is not a whole number.

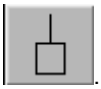
Modification of the number of significant digits
If your digit includes decimals, you can choose the number of digits after the points by ticking **Delete non significant zero** and then, by entering the number of decimals (truncated figures).

Once you defined all these parameters, click on **Next** to validate. The dialog box **First kind of ticks** appears on the screen.

Graduation parameters

Once you've defined the labels and validated by clicking on **Next**, the dialog box **First kind of ticks** is displayed on the screen. This dialog box allows you to parameterize the shape, the height, the width and the position (in relation to the support drawing) of the first kind of parameters (red ticks).

Shape of the first graduations

Select the shape of the first kind of graduations by clicking on , or  .

Dimensions of the first graduations

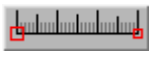
Enter the height and the width of the first kind of graduations in the two fields containing values in **mm**.

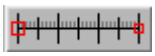
Defining the number of the first graduations

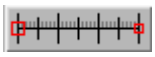
Number of spaces
There is one space less than the number of graduations. Enter the desired number. Click then in the other field (Number of ticks) to update immediately the corresponding number of graduations.

Number of ticks
There is one tick (graduation) more than the number of spaces. Enter the desired number. Click then in the other field (Number of spaces) to update immediately the corresponding number of spaces.

Position of the first kind of graduations

 Activate this option to position your ticks (graduations) above the original shape.

 Activate this option to center the ticks (graduations) on the original shape.

 Activate this option to position the ticks (graduations) under the original shape.

Remark

The blue number on the right corner at the bottom of the dialog box First kind of ticks **corresponds to the parameterizing level of the different kinds of ticks, you may want to use. In the example shown above, we have chosen to use only one kind of tick. In case we would have liked to use more than one kind of ticks, we should have filled in as many** First kind of ticks **dialog boxes as used kinds of ticks.**

SELECTING AN OBJECT OR A GROUP OF OBJECTS

Selecting an object

Selection tool

With this tool, you can select, move, resize, scale, or rotate an object.

To select an object, click on any point on its shape.
The selected shape is displayed in red with handles.

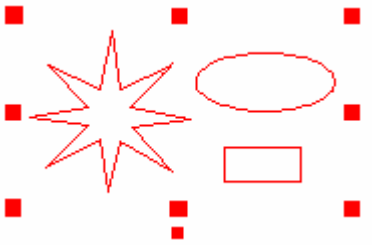
Selecting a group of objects

To select several objects, move the cursor above and outside of the first object. Click and drag the mouse towards and outside the final object, then release the button.

Remark

Check that all the objects to be selected are completely included in the window.

Handles appear around all selected objects. Click on one shape to move all of them together.



Click on the tracing of each of the objects while pressing the **Ctrl** key = Multiple selection (one by one).

Click on the tracing of one selected object while pressing the **Ctrl** key = De-selection

COPYING AND DUPLICATING OBJECTS

Copy / Paste objects

Copy
Select the object to be copied, then click on the icon.

Remark
Before pasting the object and if you are working in the same document, remember to deselect the objects to be duplicated.

Quick tip... Ctrl + C

Then, click on the **Paste** icon.
If working on the same drawing, the copied object will be exactly on the original.

If you wish to paste the object on another page, open the document and click on the **Paste** icon.
The copied object will be positioned at the same coordinates as the original.

Quick tip... Ctrl + V

Remark
To see both objects, under the View **menu, choose** Redraw **and move the second object to the desired position.**

Quick tip... Ctrl + W

Cut / Paste

Cut
Select the object and click on the icon (the object disappears).
Open the file where you want to insert it.

Click on **Paste** icon
The copy will be positioned at the same coordinates as the original.

Remark
You can find these functions Copy**, Cut, and** Paste **in the** Edit **menu**.

Copy or Cut / Paste from another software

You can also copy an object issued from another graphic or word processing software.
Select and copy the object, open the document in which you want to insert the copy and click on the **Paste** icon .

Automatic duplication

Duplication tools
4 different duplication modes are available: one by one, in rows, on a circle, or on a curve.

Duplicating one object
Select the object and then the icon. The duplicating symbol appears close to the cursor.

Click on the contour of the object to be duplicated and drag the mouse towards the desired position. Release the button. Repeat this step as many times as necessary.

Ctrl To move the copy on one axis.

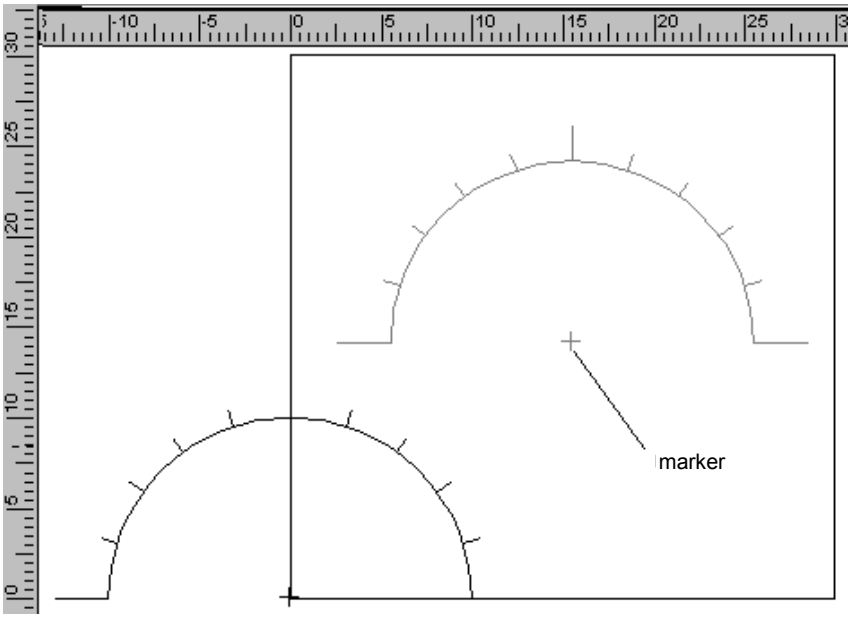
F2 to enter an accurate position of the copy. This function can be associated with the snapping mode of **F3**.

Example: duplication with a reference marker.

Select a reference marker in your drawing and position the marker.
*Press the **F3** key and select the option **Marker**.*

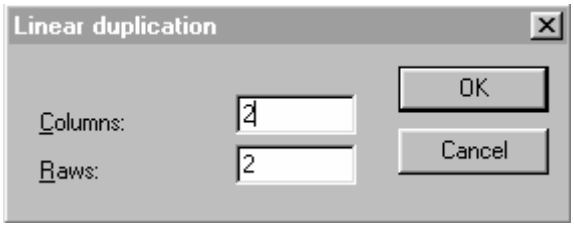
Select the whole drawing (the marker included) and click on the duplication icon.
Position the cursor of the mouse on the marker and start to move your drawing on the screen.
Check that a circle is drawn close to the cursor. This means that you are in the magnetization field of the marker.

*Without releasing the button of the mouse, press the **F2** key. Enter in the window the desired position for the reference point of the object. For our example, the **X** and **Y** values are **0**.*



Linear duplication

Linear duplication
To duplicate the object several times along rows and/or columns, select the object to be duplicated and click on the icon.

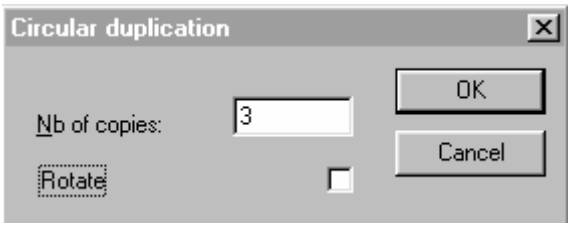


In the dialog box, enter the number of columns and rows.

Click on the object to be duplicated and drag the cursor. The copies are displayed with dotted lines. Release the button once you have reached the desired positions.

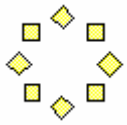
Circular duplication

Circular duplication
It is the same procedure as the linear duplication. Select the object and then the icon.

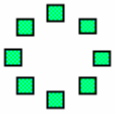


Enter the number of copies in the dialog box.

Click on the contour of the object to be duplicated and drag the cursor.
The copies are displayed in dotted lines. Release the button once you have reached the desired positions.



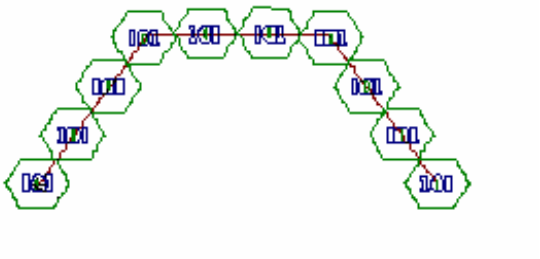
With the **Rotate** option
active: the object follows the rotation angle



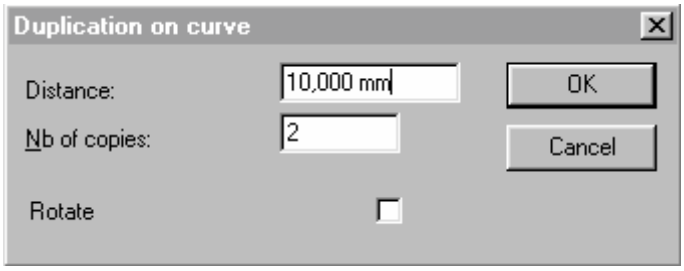
Inactive: the objects stay aligned

Duplicating an item along a curve

Duplicating an item along a curve
Position the object to be duplicated on the shape at the starting point of the duplication.



Then select the object and the curve, and click on the icon.
Be careful to respect this order of selection.

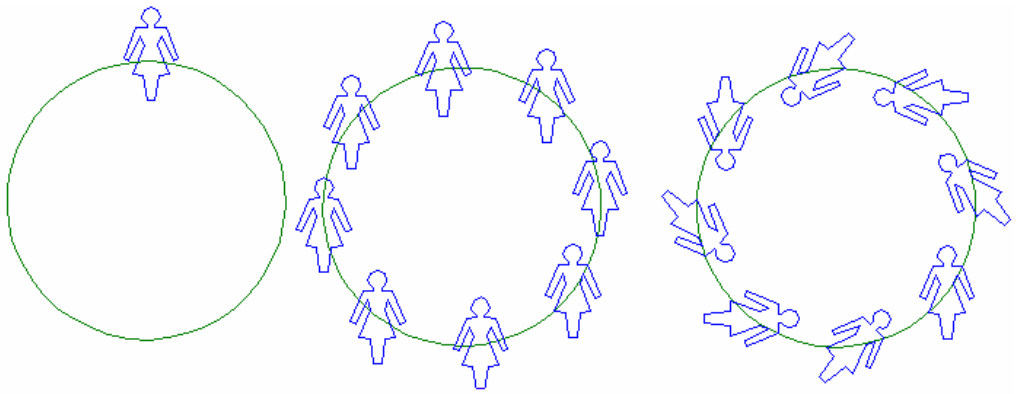


There are two ways of duplicating:

- giving the distance between each copy,
- giving the number of copies.

Rotation
This option allows you to rotate the duplicated item.

Example:



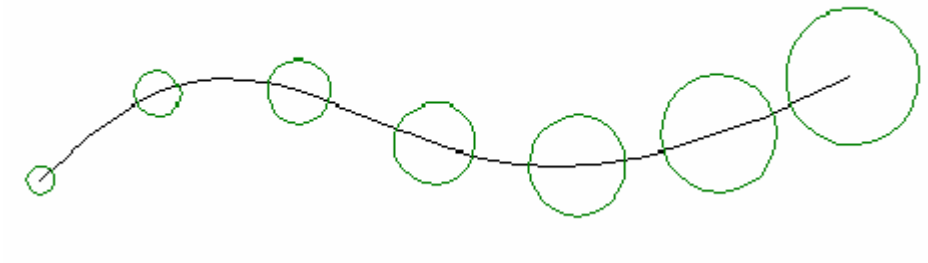
Original drawing *Simple Duplication* *Duplication with rotation*

Duplicating by distortion of an object on a curve

Draw the curve along which you want to execute the duplication and place the objects to be duplicated at each extremity of the curve.
Select the objects, then the curve, and click on the icon.

In the window **Duplication on curve**, enter the number of copies required.

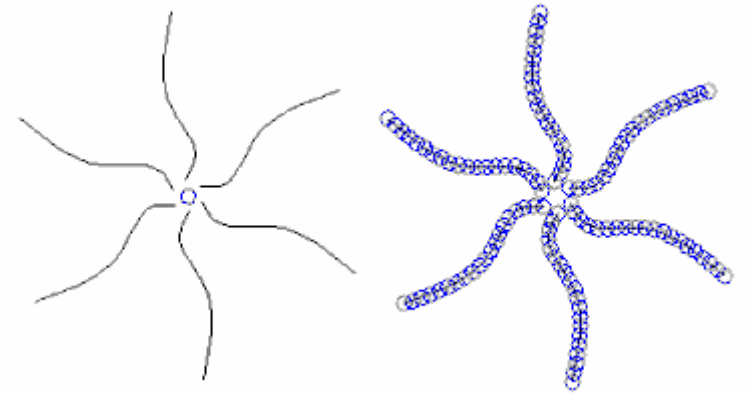
Original drawing



Result achieved with 7 copies

Simultaneous duplication on several curves

In order to duplicate an object simultaneously on several curves, you have first to combine all curves. Then, select the object to be duplicated and next the curves. Click on the icon . Indicate the desired number of copies or the distance between copies in the dialog box **Duplication on a curve**.



DELETING OBJECTS

Deleting an object

Select the object, and in the menu **Edit**, choose **Delete**.

Quick Tip ...
Select the object and use the Del **key.**

POSITIONING OBJECTS

Moving with the mouse

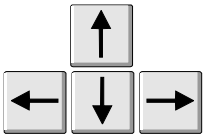
Select one or several items, using the selection functions described in the section **Selecting an object or a group of objects**, position the cursor on a contour, and while holding down the left button, drag the object towards the desired position. Release the button when done.

Move while pressing the **Ctrl** key = horizontal and vertical displacement.

Moving with F2 key

To move an object accurately, start to displace it with the mouse and click on **F2**.
In the dialog box, give the distance of the displacement on each axis with regards to the current position.

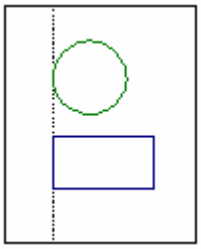
Moving with the keyboard



The arrows keys allow you to move the selected item horizontally or vertically.
The value of each move is given in the menu **Edit → Option: Arrow move**.

Aligning against a guideline or grid

Grids and Rulers are at your disposal to help you to accurately align objects.



These two functions are associated with the magnetization option, which (when activated with the **F3** key) forces objects to be aligned against the grid or the ruler.

Use of guidelines

Displaying the rulers

If the rulers are not displayed, under the **Edit** menu select **Options…**. The dialog box **Options** is displayed on the screen. Choose **Display** and select **Show rulers**.

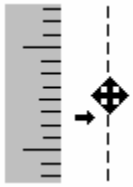
Placing a guideline

Click on one of the rulers and keeping the left button of the mouse pressed, drag the cursor towards the active area to the position desired.

If you click on a vertical ruler you have a vertical guideline, and if you click on an horizontal ruler you have a horizontal guideline.

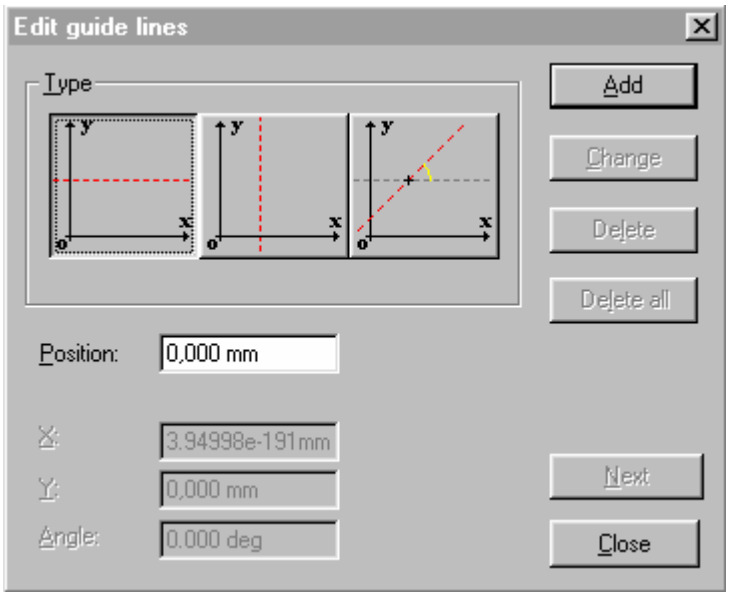
Remark

You can numerically position a guideline using F2**.**
Click on one of the rulers and without releasing the left button of the mouse, drag the cursor towards the active area. Press the F2 **key, enter the position of the guideline and validate.**



You can have an unlimited number of guidelines.

To delete, add, or move guidelines, go to **Change → Guide layer...**
The modification of guidelines dialog box allows for accurate positioning of the guidelines.



You can also add a guideline by choosing the axis and the position. Then select **Add**.

To move a guideline, enter the new numerical values in its field and click on **Change**.

Quick tip...

Double-clicking on a guideline will display the Edit guide lines **dialog box.**

Snap on guidelines

Press the **F3** button and select: **Grid and Guide**.
As soon as the displaced object enters into the magnetization field of the guideline, it is snapped onto it.

Magnetization field

You can reduce or increase the magnetization field in the menu **Edit → Options…**. The dialog box **Options** is displayed. Choose **Display** and enter the numerical value of the magnetization field in **Snap distance**.
The **default value** is [8].

Aligning against a grid

Like the guidelines, the grid helps in the positioning of objects. The grid creates magnetic points with an adjustable distance.

Displaying the grid

To display the grid, go to **Change → Grid** and select **Visible**.
The default value is 100 mm on the 3 axes and the point of origin is 0.

Modification of the parameters of the grid

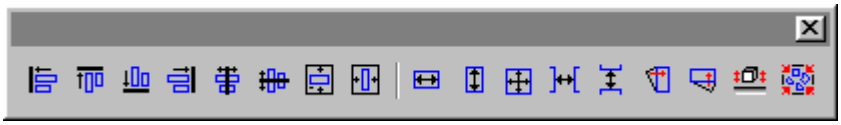
The grid is useful for repetitive dispositions of objects at equal distances. You can define the parameters in accordance with your needs and with the type of work undertaken.

Magnetization of the grid

When you have finished defining the parameters of the grid points, select the **Enabled** option so that the grid becomes magnetized and **Visible**, then select one by one the objects and position them against the desired points of the grid.

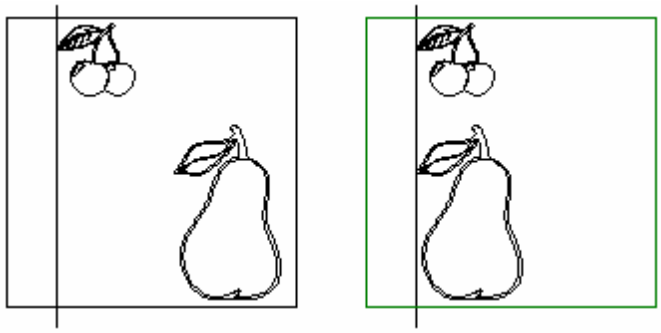
Positioning with special tools

Alignment tools
The procedure is the same for all the alignment modes (on the left, on the right, at the top, at the bottom). First select the object(s) you want to align with **Ctrl** key, and then choose the corresponding icon.

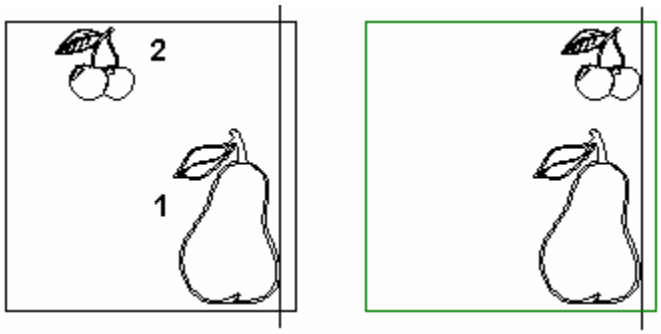


Aligning

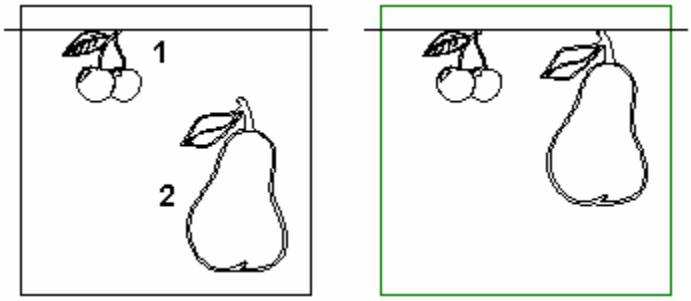
Align on left
All the objects will be aligned according to the X min. value of the first selected object.



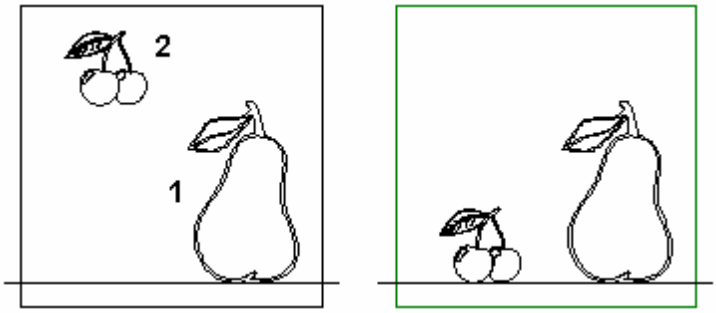
Align on right
All the objects will be aligned according to the X max. value of the first selected object.



Align to top
All the objects will be aligned according to the Y max. value of the first selected object.



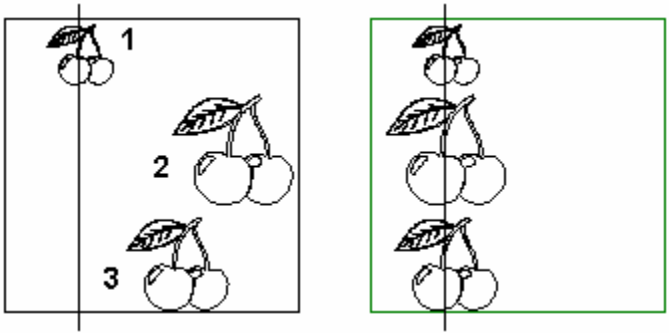
Align to bottom
All the objects will be aligned according to the Y min. value of the first selected object.



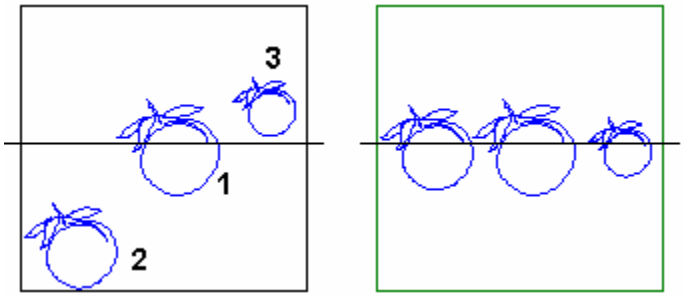
Vertical and horizontal aligning

To align the objects, it is sufficient to define the axis to which you want to align them.

Vertical aligning of the objects
The objects will be vertically aligned based on the center of the first selected object.



Horizontal aligning of the objects
The objects will be horizontally aligned based on the center of the first selected object.

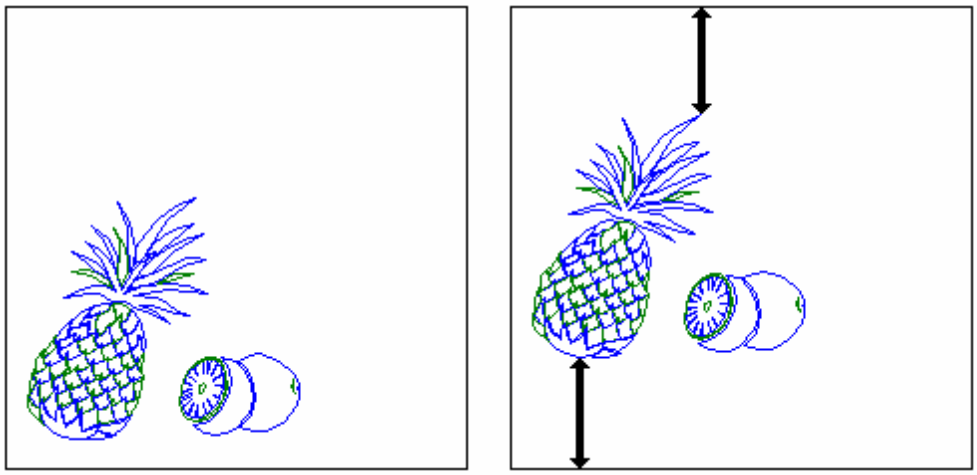


Remark
To reach a perfect aligning, vertically and horizontally, use these two functions one after the other.

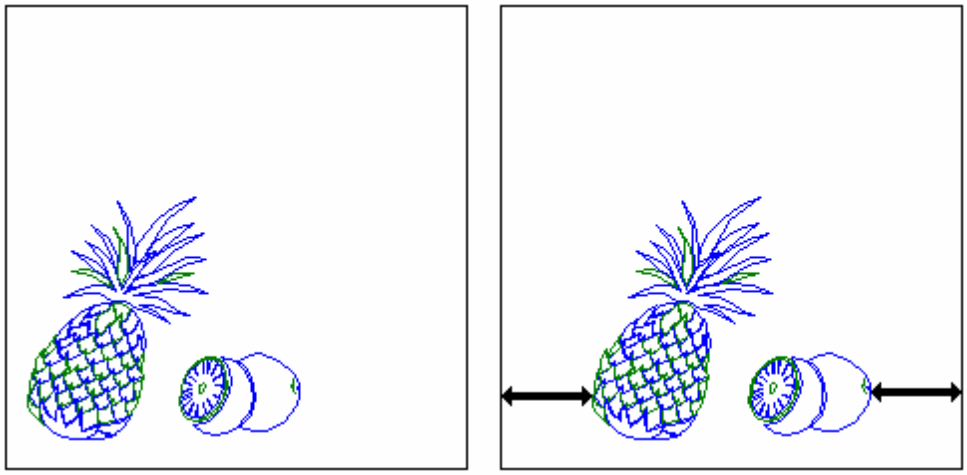
Vertical and horizontal centering of the objects

You can place the objects horizontally or vertically at equal distances from the borders of the active area.
The spacing created will be an average between the closest object to the vertical or horizontal borders of the active area (depending on the type of alignment desired).

Vertical center



Horizontal center

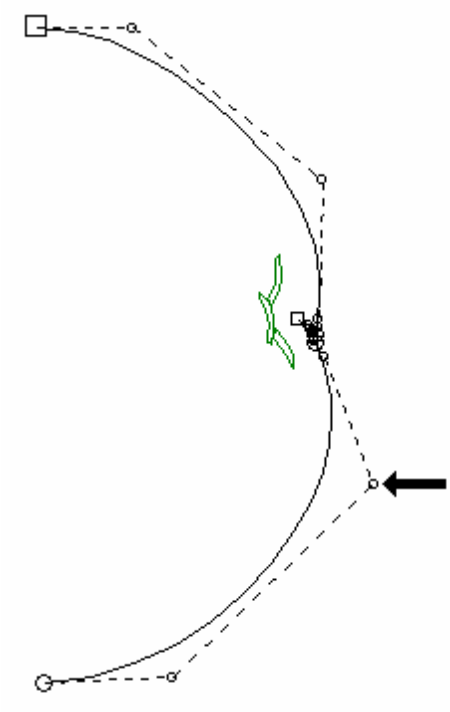


Remark

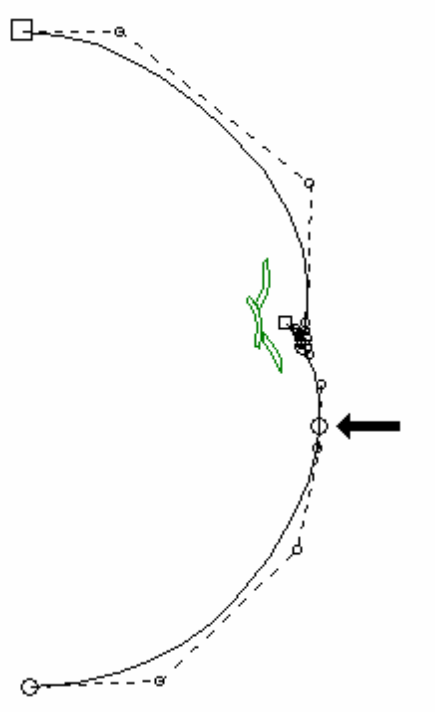
To obtain an equal distance from each border, check that the points outside of the curve are not outside the tracing as the function will give a different global size compared to the real dimensions of the objects.

You just need to add a point on the curve to avoid this phenomenon.

Example



The point outside of the curve is outside the tracing.
The global size is more important compared to the real dimensions of the drawing.

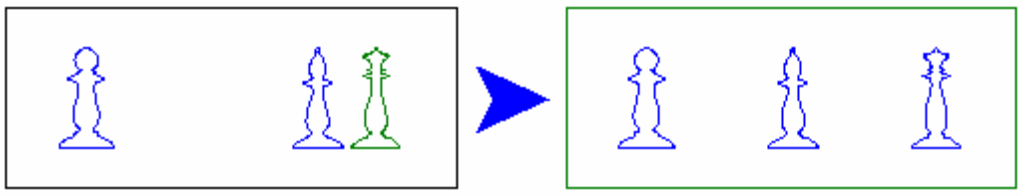


There is no point outside of the curve.
The global size corresponds to the real size. of the drawing

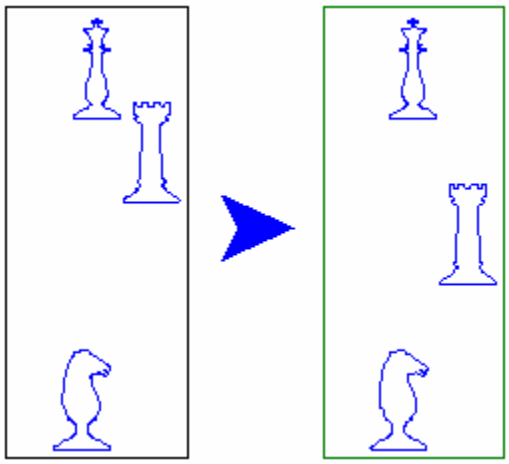
Automatic positioning

An automatic positioning can be done to get an equal distance, vertically or horizontally, between several objects. Just select the objects and click on the corresponding icon.
A medium spacing is automatically generated and the occupied space of the group is not modified.

Horizontal spacing

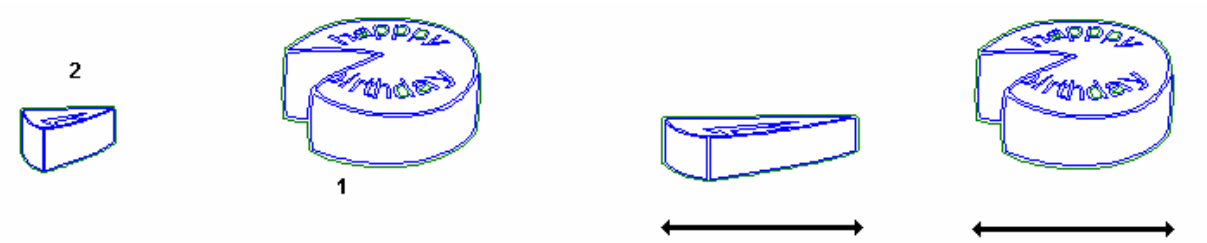


Vertical spacing

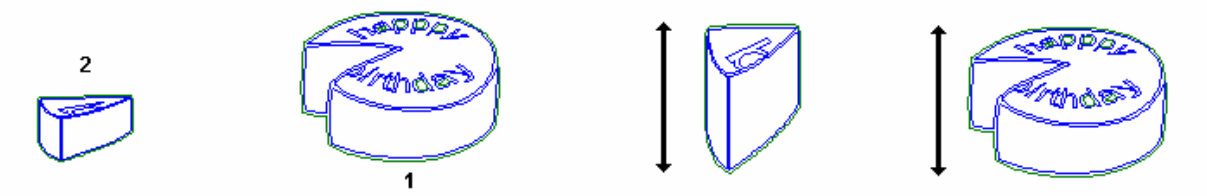


Automatic resizing

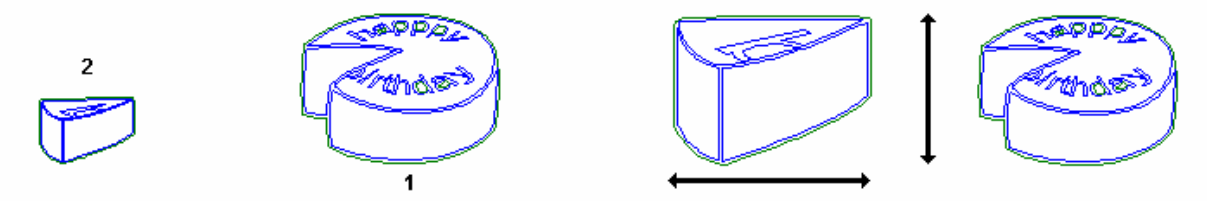
Automatic horizontal resizing
The first selected object gives its horizontal length to all of the other objects.



Automatic vertical resizing
The first selected object gives its vertical length to all of the other objects.



Automatic full resizing
The first selected object gives its horizontal and vertical lengths to all of the other objects.

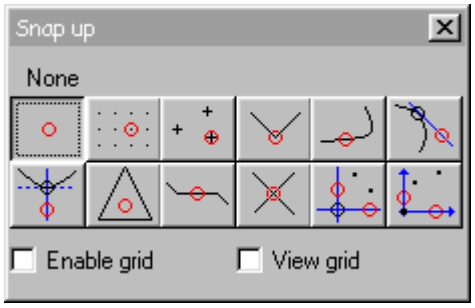


Aligning along a vertical or horizontal

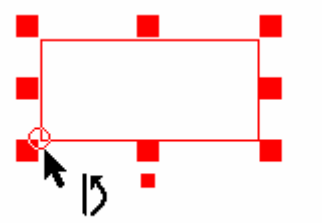
Aligning along a vertical axis

Using this function allows to make an object rotate to align along the vertical axis.
Select the object and strike the **F3** key to choose a snap mode.

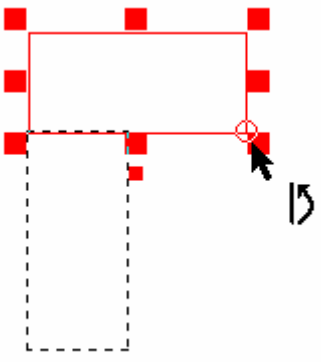
The dialog box **Snap up** is displayed. Select the option **Control point** symbolized by the button and validate by clicking on **OK** let's quote that the name of each option activated is written above the corresponding button).



Click on the icon to make work the function **Aligning along a vertical axis**.



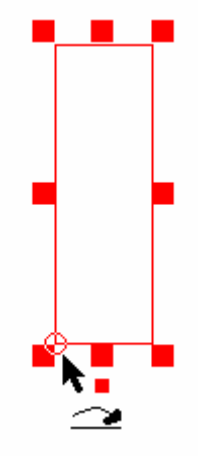
Select the point around which you want to make the object's rotation to align it along the vertical axis.
Select then the opposite point on the horizontal axis. This opposite point is automatically aligned on the vertical axis. The position of the object displayed with a dotted line shows that the operation went well.



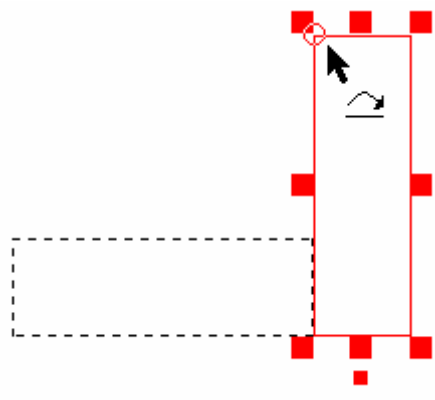
Aligning along an horizontal axis

Using this function allows to make an object rotate to align along the horizontal axis.
Select the object and strike the **F3** key to choose a snap mode. The dialog box **Snap up** is displayed. Select the option **Control point** and validate by clicking on **OK**.

Click on the icon to make work the function **Aligning along an horizontal axis**.



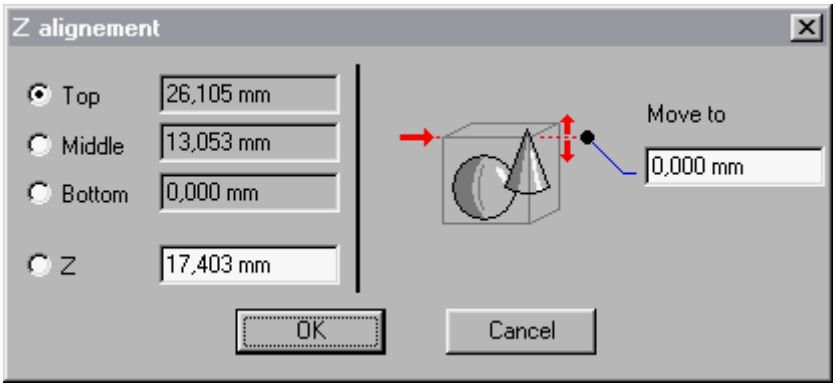
Select the point around which you want to make the object's rotation to align it along the horizontal axis.
Select then the opposite point on the vertical axis. This opposite point is automatically aligned on the horizontal axis. The position of the object displayed with a dotted line shows that the operation went well.



Z positioning

This new function allows you to reposition on the **Z-axis** any kind of 2D or 3D object (shape or surface made in the CAD module, imported or digitized surface or TypeArt-object). One of the greatest interests of this function is the repositioning of a TypeArt-object before sending it to the machining.

Select the object to be repositioned on the Z-axis and click on the icon **Z positioning** . The dialog box **Z alignment** appears on the screen.



Select in the left part of the dialog box the reference point of the object to be positioned. Enter then the new Z value in the field **Move to** (right part of the dialog box).

Top
The top of the object will be moved to the height defined in the field **Move to**.

Middle
The middle of the object will be moved to the height defined in the field **Move to**.

Bottom
The bottom of the object will be moved to the height defined in the field **Move to**.

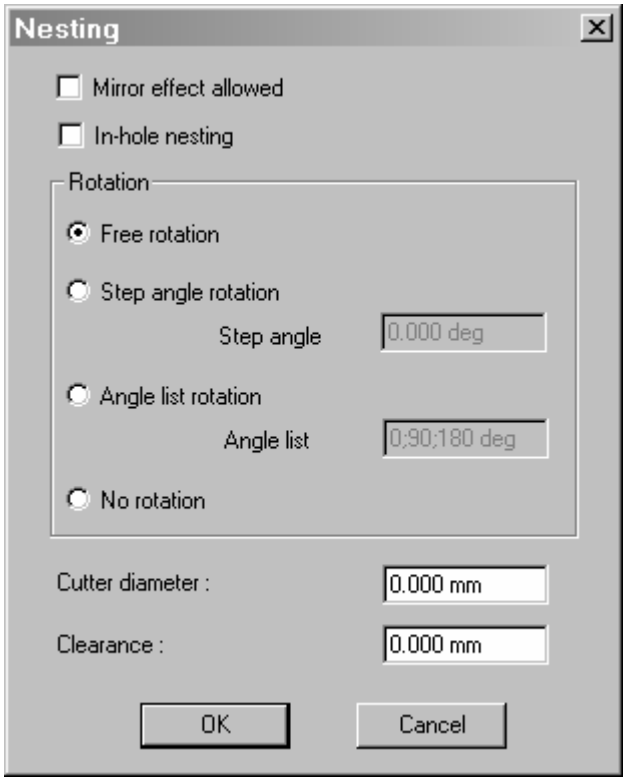
Z
The point of the object with the height defined in that field will be moved **Move to**.

Nesting

This function computerizes an automatic layout of objects in a simple bounding box (a contour) or in a bounding box with holes (one contour with contours combined inside). The room is optimized in order to save material.

Select the bounding box, then all contours to be nested in the bounding box. The selected elements have to be closed contours (neither opened curves, nor text with single-line font). Click on the icon **Nesting** (**Alignment** toolbar).

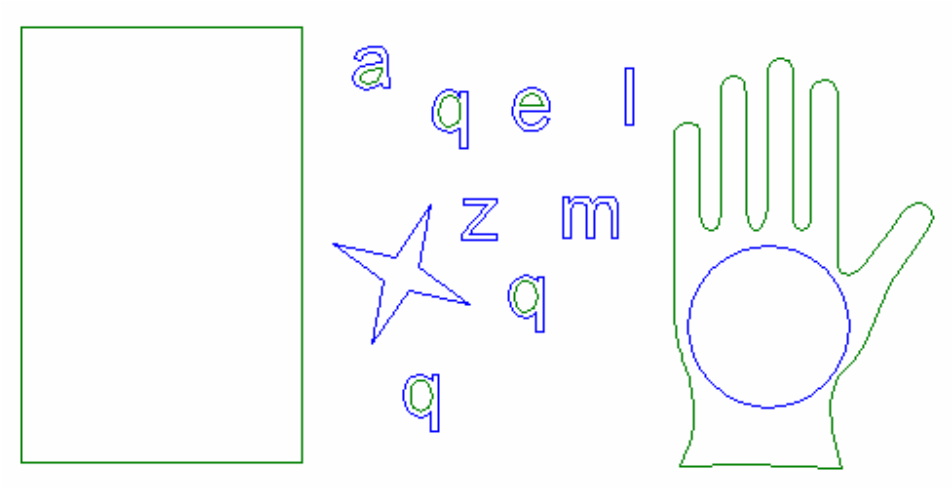
The following dialog box is displayed on the screen:



Mirror effect allowed
Enables the rotation of the object in a mirror when needed. It requires that the objects haven't any distinctive sides.

In-hole nesting
The remaining material of each object can be used to nest another object inside the hole.

Example



Objects to be nested

Result of the nesting by filling the holes

Rotation

Free rotation
Optimized nesting without any rotation angle to be respected.

Step angle rotation
Optimized nesting with indication of a precise angle of rotation.
Enter a step angle: multiples of this angle are accepted (for instance 90°: step angles 0°, 90°, 180° and 270°).

Angle list rotation
Optimized nesting with two precise angles of rotation or more to be respected.
Enter the angle values separated by a semicolon.

No rotation
Optimized angle without any rotation angle.

Cutter diameter
Enter the value of the tool diameter used for the machining.

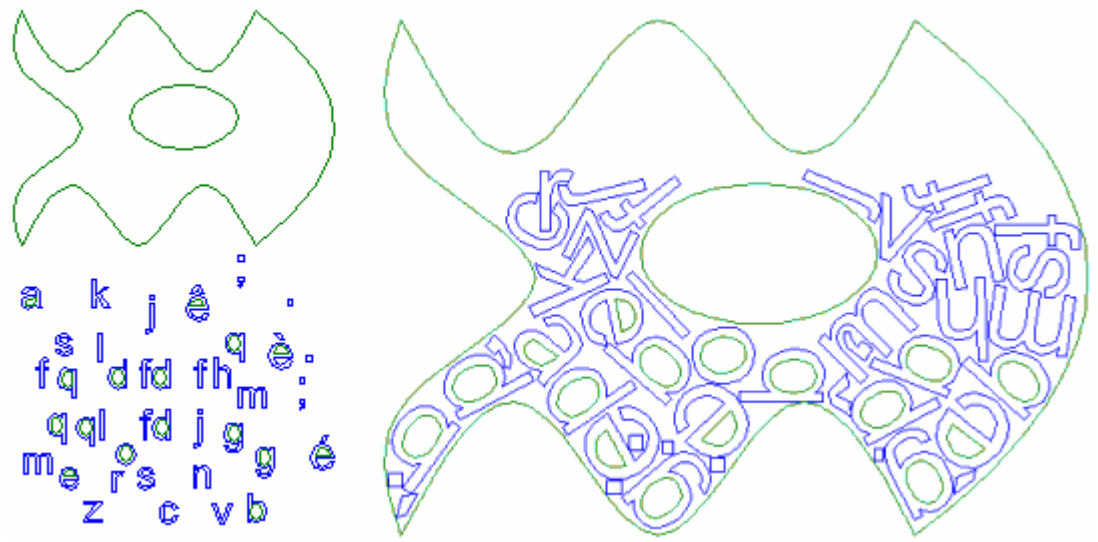
Clearance
Enter the value of the desired additional distance between objects (includes the tool diameter).

NOTES

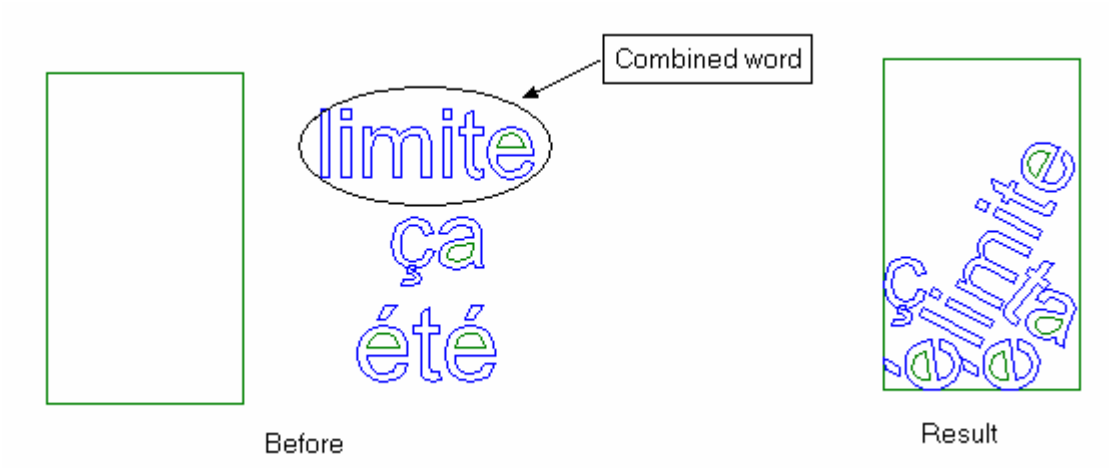
The resulting distance between nested objects is not equal to the "Cutter diameter" + the "Clearance". A distance "delta" is automatically added by the optimization calculation.
The Nesting function allows to keep the relatives positions of the letters between them. If you wish to keep the entire word, select and combine it. Accents and other diacritic marks are still combined to the letter they are associated with. Tip the text and convert it into curves. A letter with an accent is grouped. If you desire to nest a letter separated from its accent, select the whole thing and click on the icon

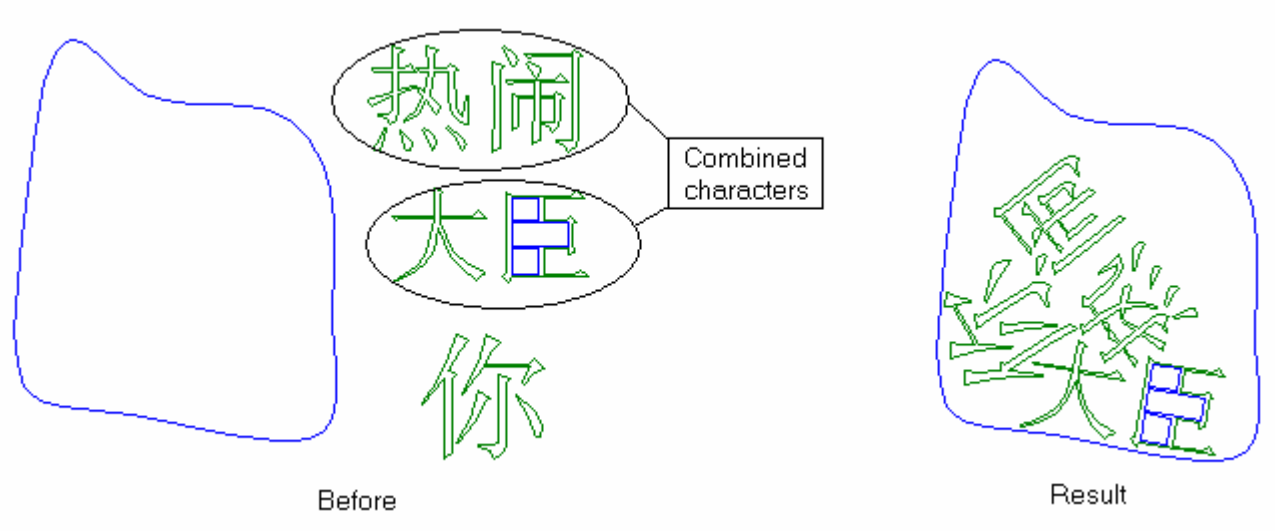
Uncombine .

Examples



Nesting of letters with uncombined diacritic marks




热闹
大臣
你
Combined characters
Before
Result

TRANSFORMING OBJECTS

Transformation tools

Transformation tools

In this section, you will first learn how to scale, stretch, mirror, rotate, and create an envelope. Then, you will learn how to manipulate shapes; move, add, and delete control points; to disconnect or weld a contour; and to combine or uncombine several contours.

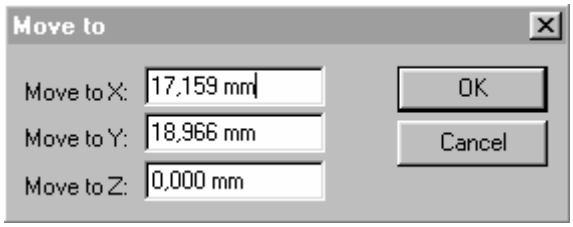
Moving an object

Move object

Select the object to be moved and click on the **Moving tools** icon. The symbol for displacement will appear close to the cursor. Drag the mouse to displace the object and release the button to validate the position.

Ctrl = Displacement on one axis

F2 for moving the object with accuracy. Therefore, select the object and place the icon anywhere on it.
Click on the left button of the mouse, keep it pressed, start to move the object and strike the **F2** key. Enter the desired values in the dialog box as shown below.



This function can be used associated to a snapping mode of **F3**.

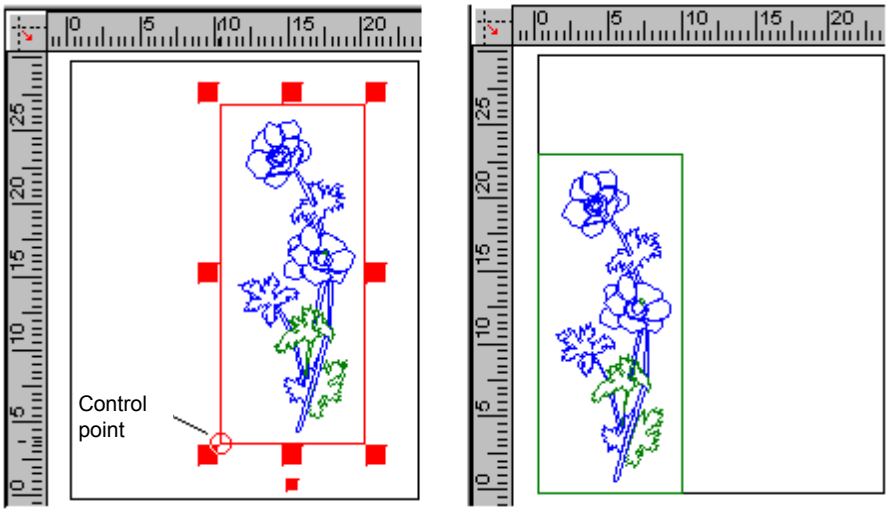
Example: moving using the control point

*Press the **F3** key and select the snapping mode **Control point**..*

*Select the whole drawing and click on the **Move object** icon.*
Position the cursor of the mouse on the desired control point and start the displacement on the screen.
Check that a circle appears near the cursor of the mouse. It means that you are in the magnetic field of the control point.

*Without releasing the button of the mouse, press the **F2** key. Enter in the window, the desired position of the reference point of the object.*

For this example, the X and Y values are 0.



Scaling

With the mouse

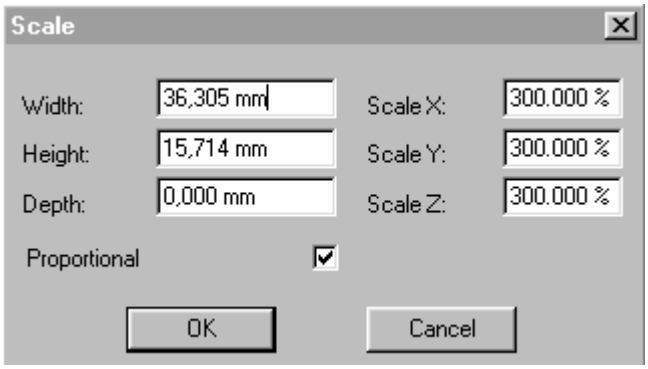
You can scale an object with its handles or with the appropriate icon.

Shift	= 2 sides at the same time
Shift + **F2**	= 2 sides at the same time with an accurate scale
Ctrl	= To obtain steps of 100 by 100%.

Select the object and click on a handle in a corner.
The cursor changes to a sizing pointer; drag the cursor in the desired direction to obtain the required size.
Then release the button.

Accurate scaling

With F2 key
Select the object and click on a handle. The cursor changes to a sizing pointer. Press the **F2** key and give the numerical scaling factor or the exact dimension desired in the fields of the selected axis.



Scaling tool

Scale
After having selected the item and this icon, click anywhere in the active area and drag the cursor: the transformation is centered. Release the button on the mouse when done.

Ctrl = To obtain steps of 100 by 100%.

Vertical and horizontal stretching

You can use the handles or the Stretching icon.

With the mouse

Select the object and click on a lateral handle.
The cursor changes to a sizing pointer. Drag the cursor. A dotted frame which will follow the movements of the cursor, appears around the object.
Release the button of the mouse when the object has the desired shape.

Accurate scaling

Select the object and click on a lateral handle.
The cursor changes to a sizing pointer; press the **F2** key and give the numerical scaling factor or the exact dimension desired in the field of the selected axis.

Shift	= 2 sides at the same time
Shift + **F2**	= 2 sides at the same time with an accurate stretching
Ctrl	= To obtain steps of 100 by 100%.

Stretching tool

Stretch
After having selected the object and this icon, click anywhere in the active area and drag the cursor.
The stretch is done by the center, and it can be vertical or horizontal.

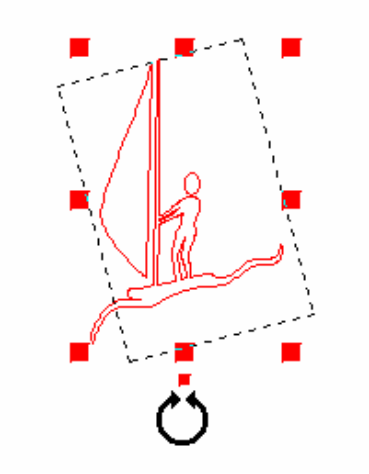
Shift = 2 sides at the same time

Rotating an object

With the mouse

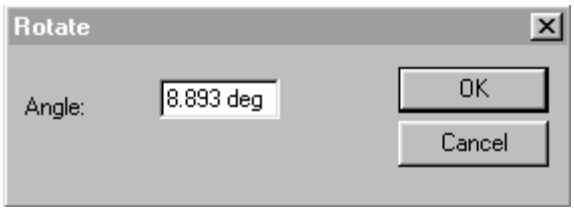
You can rotate an object with the appropriate tool or with the mouse by clicking the little handle below the selected area.

The cursor changes into a rotating pointer.



Hit: F2 To give numerical values to the angle
Ctrl Rotation by 15° at a time.

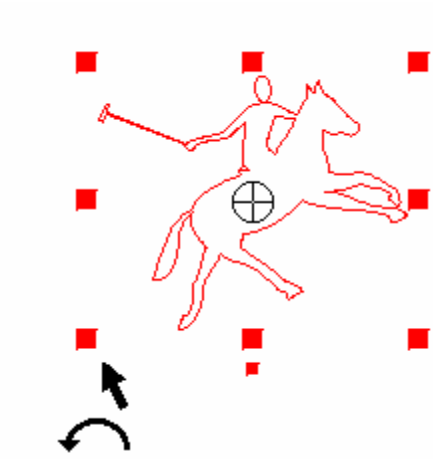
To enter numerically the angle value for the rotation, start rotating as described above and strike the **F2** key. Enter the value in the dialog box **Rotation** displayed on the screen.



Rotating tool

Rotation

Choose a handle and turn the object around its center of rotation in the desired direction. The dotted box outlining the item represents the object during this action. Release the button of the mouse when the position corresponds to the desired angle.
To know the angle of rotation during the operation, check the bottom status line.



Moving the pivot point

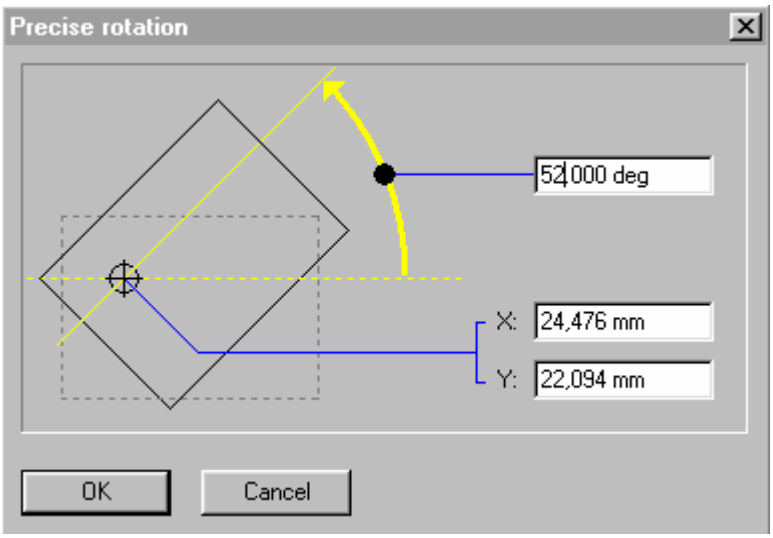
The special handle shown in the center of the object is the pivot point. You can move this center where you want by displacing the pivot point. Only the center point will be moved.

Then you can pivot the object around its new pivot point by dragging one of the corners of the selected perimeter.

With **F2** and **F3** = accurate positioning of the pivot point.

Numerically rotating and moving the pivot point

Select the object and click on the **Rotation** icon. The dialog box **Precise rotation** is displayed on the screen.

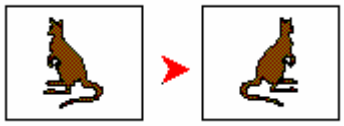


Enter the rotation angle in the upper field of the dialog box. If the pivot point has to be moved, enter its new coordinates **X** and **Y** in the appropriate fields. Click on **OK** to validate.

Mirror of an object

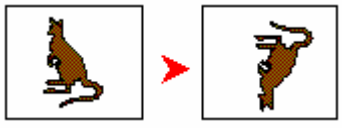
Horizontal mirror

Select the object and click on the icon **Horizontal mirror**.
The object rotates around a centered vertical axis.



Vertical mirror

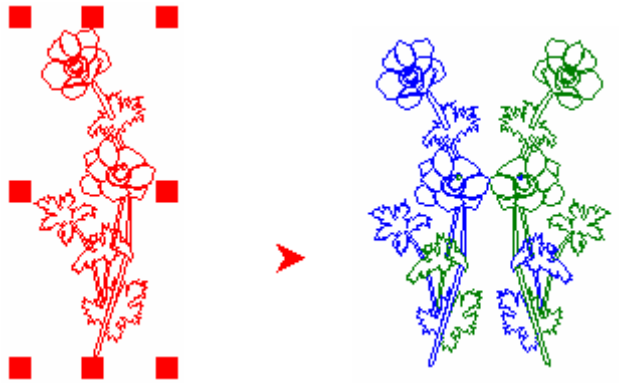
Select the object and click on the icon **Vertical mirror**.
The object rotates around a centered horizontal axis.



Remarks:

1. **You can keep the original picture by striking the** Shift **while clicking on the** Mirror **icon.**
2. **You can modify the rotation axis by striking the** Ctrl **key while clicking on the** Mirror **icon. The rotation axis is not centered anymore: it is on the right side (concerning the horizontal mirror) or on the base (regarding the vertical mirror) of the selection window.**

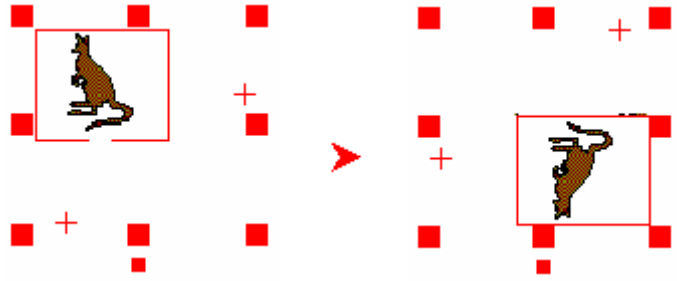
*Example with an Horizontal mirror application (in this case, both keys **Shift** and **Ctrl** are pressed while clicking on the icon):*



Original *Result*

Mirror around an axis

Define an axis by inserting two markers on the material. Strike the **Ctrl** key and keep it pressed to select the object, and then the two markers. Click then on the **Mirror around an axis** icon. The object rotates around the axis you defined.



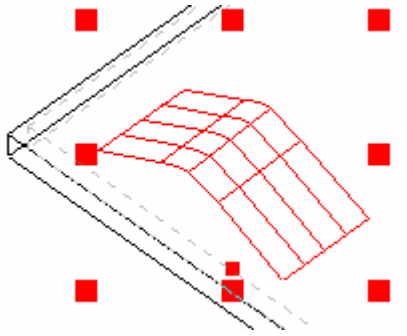
Remark

You can apply the Mirror **function to a text. Therefore, you need to convert it into curves.**

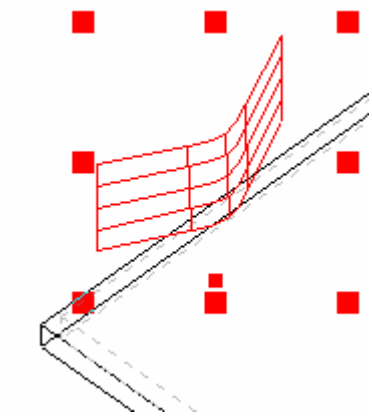
Permuting plans

Flip plan

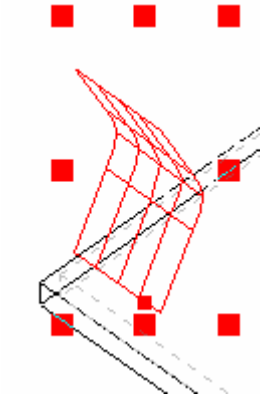
You can simply reposition the object you created (this function is very useful for the surfaces) with this tool.
To make flip a plan, select the object.



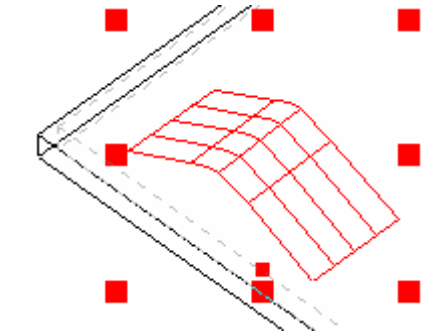
Click a first time on the **Flip plan** icon to invert the coordinates **X**, **Y** and **Z** as shown below:
X becomes **Z**,
Y becomes **X**,
Z becomes **Y**.
The consequence is the new positioning of the object (coordinates **Z, X, Y**):



Click a second time on the **Flip plan** icon to invert the coordinates **Z, X** and **Y** as shown below:
Z becomes **Y**,
X becomes **Z**,
Y becomes **X**.
The consequence is the new positioning of the object (coordinates **Y, Z, X**):



Click a third time on the **Flip plan** icon to get back to the starting coordinates **X, Y** and **Z** as shown below:
Y becomes **X**,
Z becomes **Y**,
X becomes **Z**.
The object gets back to the original position (coordinates **X, Y** and **Z**):



SPECIAL EFFECTS

Special effects tools

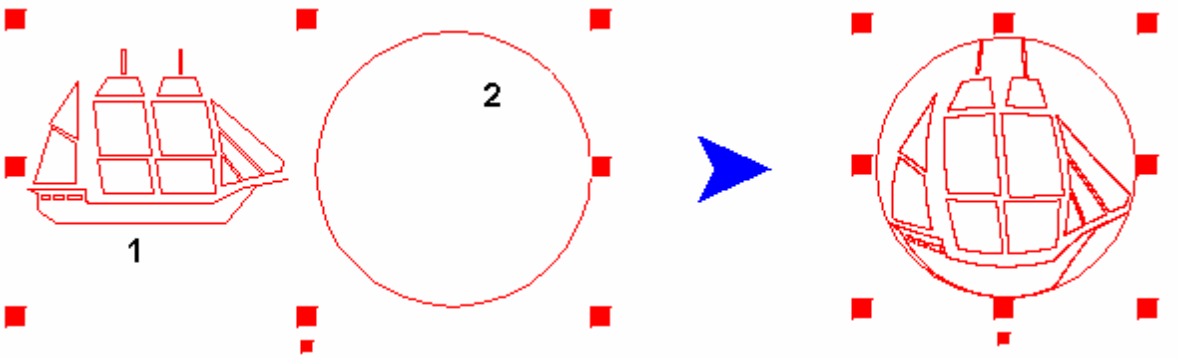
Special effects tools
The icons available in this box will allow you to transform contours, to weld them or to create a parallel contour.

Placing an object inside an envelope

Importing an envelope

There are different enveloping shapes you can choose. Click on the **Symbol** icon and in the directory **Envelope**, double-click on the desired envelope to select and to display it on the screen.

Select the object then the envelope and click on the **Envelope** icon .



Placing a text inside an envelope

Select the text and convert it into a curve with the **Convert into curve** icon .
(Tip: Group the text to select it easily afterwards).

Click on the **Symbols** icon , then select the **Envelope** directory. The different shapes are displayed. Select one of them and place it on the active area.

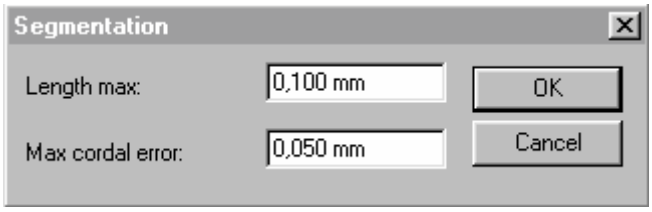
First select the text (converted into a curve), and then, without deselecting the text, select the shape with the **Ctrl** key and click on the **Envelope** icon .

Example

Segmentation

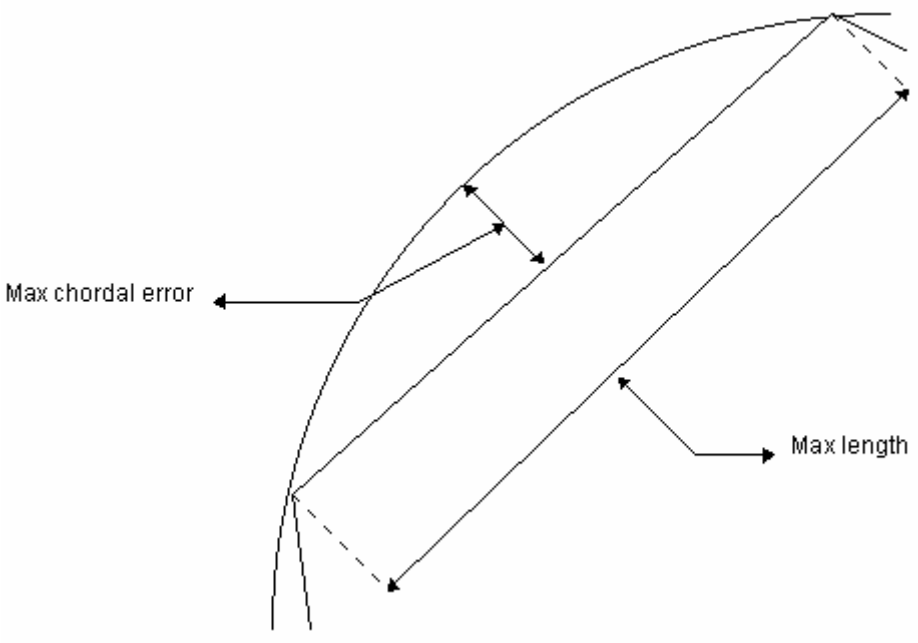
Use this function to cut a contour in segments. You can define the maximal chordal error (distance between the original contour and the created segments) as well as the segment length.

Segmentation
Select the text or the object. Click on the **Segmentation** icon. The dialog box **Segmentation** is displayed on the screen.



Enter the values in the appropriate fields.

Length max.: maximal length of the created segments.
Max chordal error: maximal distance between the original contour and the created segment.

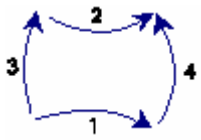


Remarks:

1. **In case of a text, convert it into curves by clicking on the corresponding icon . Otherwise you could not use the** Segmentation **function.**
2. **This function is very useful if you want to brace and map tool paths on skewed shapes. It can be used during the calculation or before it (so that you know in advance the number of created segments).**

Creating an envelope

You need to draw 4 open curves oriented as shown below to create an envelope:

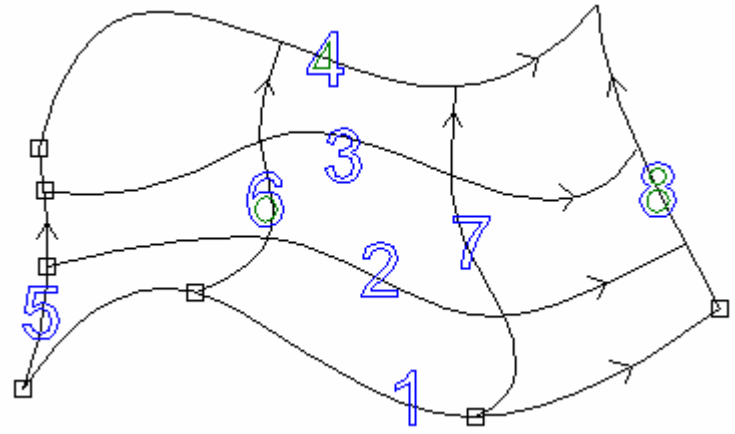


Then select them in the following order: **1 + 2 + 3 + 4.**

Combine these curves by clicking on the **Combine** icon .

Export this set with the **Syb** format by clicking on the **Export** icon or with the menu **File → Export**, and save it in the directory **Symbols → Envelop**.

You can create complex envelopes in order to customize the warping inside the envelope, using 8 curves or more, in the same way as for a simple one.



Complex envelop with representation of the curve direction and their selection order

The instructions mentioned below must be followed:

- The curves must have the same direction (vertical or horizontal).
- The curves must be linked together by using in the **Snap up** mode the **Control points** or **Along the contour** (**F3** key).
- First of all, select all horizontal curves, from the one which starting point is linked to the starting point of the first vertical curve (See figure above: first left bottom point)
- Then, combine the curves before exporting them in the symbol library (format *.syb).

Reversing the direction of contours

Reverse direction tool
With this function, the direction of the contours can be reversed.
Select the contour to be reversed and click on the icon.

Show the direction of the closed contours

The direction of the closed contours is indicated by the color: blue for clockwise contours, and green for the anti-clockwise contours.

Show the direction of the open contours

The open contours appear in **black** whatever their direction may be.

In the **Shaping mode**, you will notice that the starting point of the contour is indicated by a bigger square (open contour) or by a double square (closed contour).

Remark

The direction of the rotation can modify the machining order for the cutting or the extrusion of the material. The starting point of the contour = the starting point of the machining.

Welding

4 different ways of welding overlapping contours are available:

Join

With this function overlapping contours will be united. The achieved result is a curve assuming the attributes of the original drawing.
Select all the contours to be joined and click on the icon.

Example

Original drawing *Result*

Common

From two overlapping groups, a new object will be created by using the **Common** function. The sketching of this new object will assume the common area of these groups.

Group or combine the contours in 2 groups, select them, and click on the icon.

Example

Original drawing *Result*

Group 1: the combined stripes
Group 2: the arrow

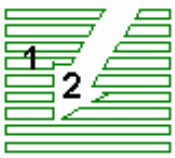
Subtract

This mode will allow you to remove one of two overlapping objects or groups of objects. Depending on the order of selection, the sketching will be different.

Examples

Original drawing:
Group 1: the combined stripes
Group 2: the arrow

Results



Order of selection: *GROUP 1 - GROUP2*

Order of selection: GROUP 2 - GROUP 1

Uncommon

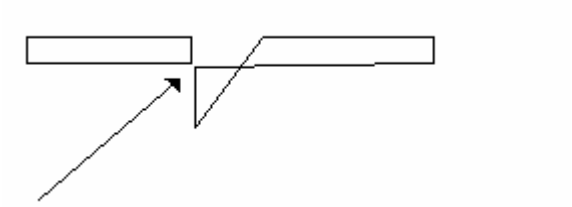
By cutting the overlapping objects with the **Uncommon** function, you will be able to separate the two objects at their intersecting point, and, therefore, create a new object.
Select the objects and click on the icon.

Example

Original drawing *Result*

At first glance, it seems as nothing has changed, but under the shaping mode, zoom on the intersecting area.

You will notice that at each intersecting contour, two cutting points are perfectly superimposed.



Separate the points by dragging one of them away from the other. This allows you to see the full effects of this mode.

Remark

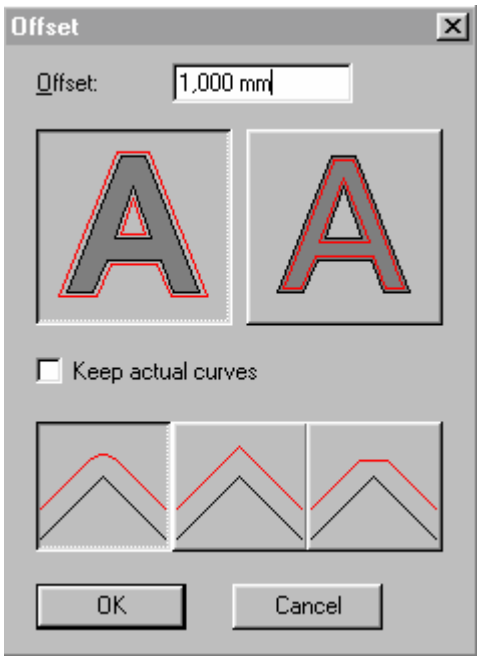
Uncombine the whole group to have independent elements.

Offset

Offset

This function automatically generates the inner or outer tracing of the selected contour.

The offset and outline angles can be defined depending on the desired result.
Select the contour or the object and click on the icon **Offset**. The dialog box **Offset** is displayed.



Remark

To maintain the original contour, select the option Keep actual curves **in the dialog box.**

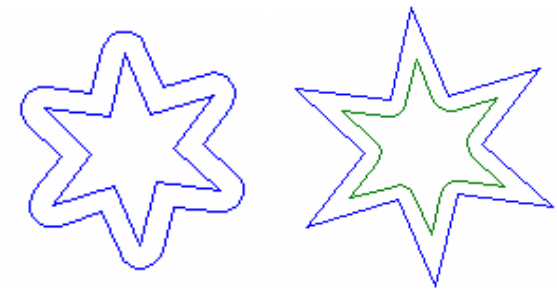
Offset parameters:

Offset
Enter the offset distance required.

Outer
The outline is executed outside the contour.

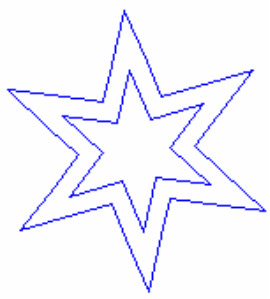
Inner
The outline is executed inside the contour.

Rounded
The angles of the outline, whether interior or exterior of the contour, are rounded.

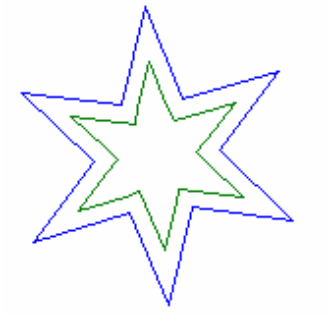


Outer *Inner*

Sharpened
The geometry of the traced angles is kept.

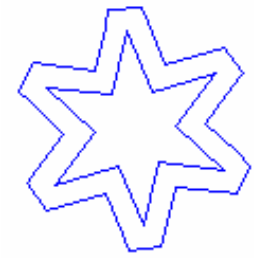


Outer

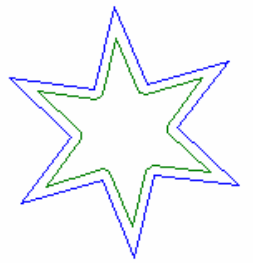


Inner

Chamfered
The traced angles are chamfered.



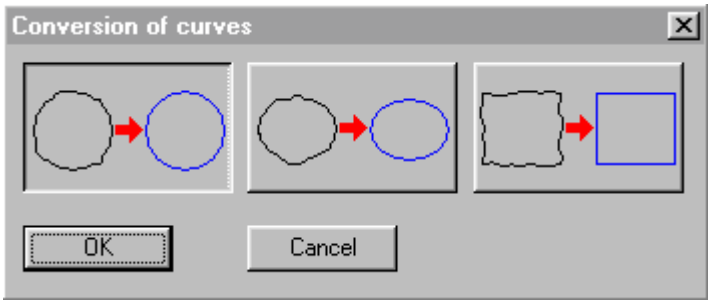
Outer



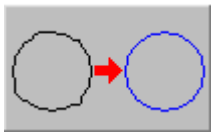
Inner

Converting curves

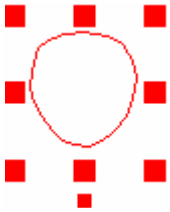
Conversion of curves
By using this function, you can transform a curve into a circle, an ellipse or a rectangle closest to the original shape. Select the shape to convert and click on the icon **Conversion of curves**. The dialog box **Conversion of curves** is displayed.



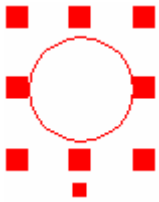
Converting a curve into a circle

To convert a curve into a circle, click on  in the dialog box **Conversion of curves**.

Example

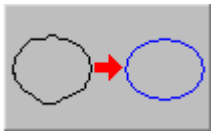


Original drawing

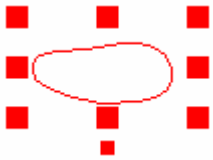


Result

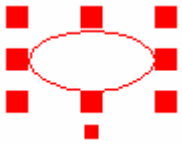
Converting a curve into an ellipse

To convert a curve into an ellipse, click on  in the dialog box **Conversion of curves**.

Example

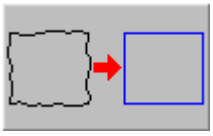


Original drawing

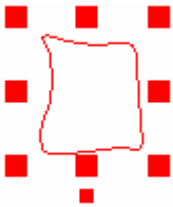


Result

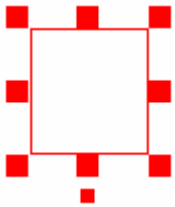
Converting a curve into a rectangle

To convert a curve into a rectangle, click on  in the dialog box **Conversion of curves**.

Example



Original drawing



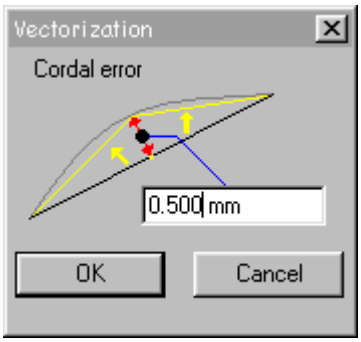
Result

Vectorizing in arcs and segments

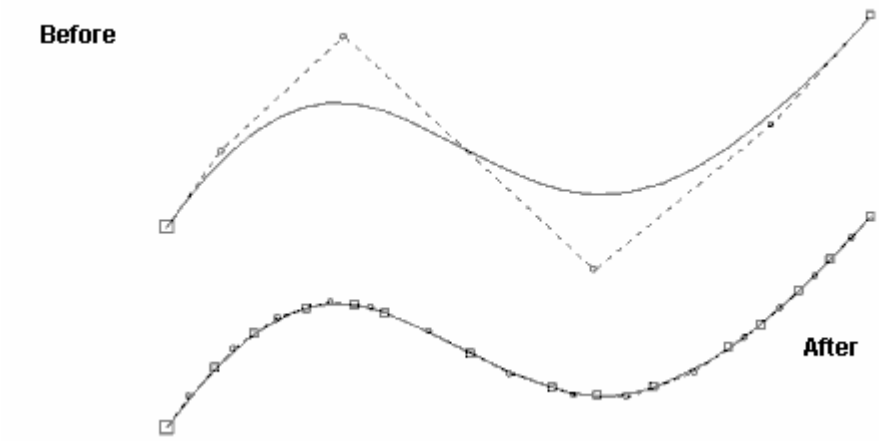
This feature is particularly useful:

- For creating toolpaths like plotting, 2D- and 3D-cutting, as far as the machine-codes are *G2-Type* (i.e. clockwise contours) or *G3-type* (i.e. anticlockwise contours) codes. This allows you to reduce drastically the number of units in the machining files.
- Before creating an offset, as far as the offset created on a contour made up of arcs and segments is also a contour made up of arcs and segments with the same number of points as the original contour.

Select the curve to be vectorized and click on the **Vectorize in arcs and segments** icon . The corresponding dialog box, in which you must enter the chordal error (i.e. the tolerance between the created segments and the original curve), is displayed on the screen. Once you defined the cordal error, validate through **OK**.



Example



Intersection markers

This function detects automatically superposed points and curves, crossed contours (intersection) and loops on the same curve. Possible problems can be avoided when toolpaths are calculated.

Select the object to be controlled and click on the icon . Intersection points are represented by markers.

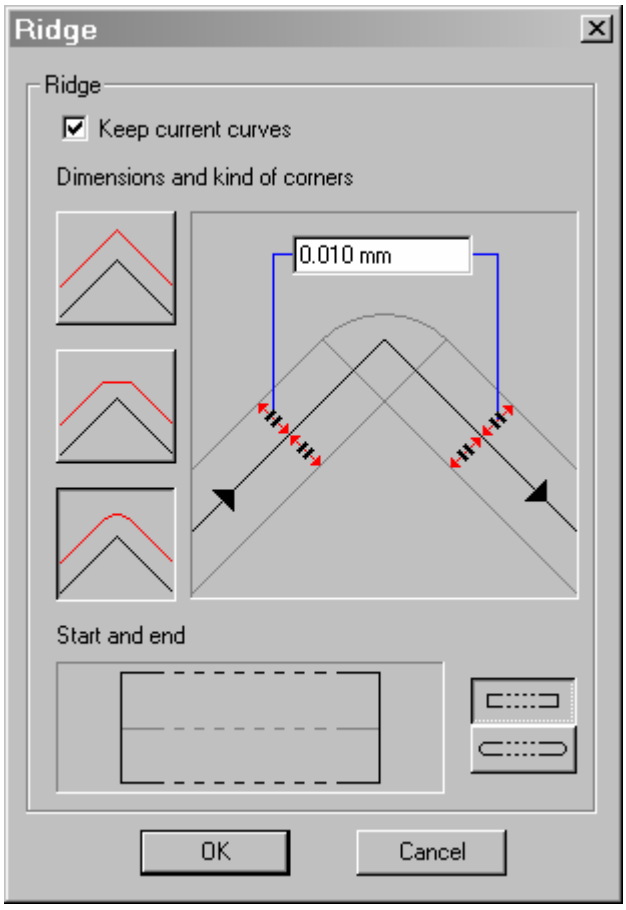
+ **CTRL** allows the automatic detection of opened contours. These are characterized by markers at each end.

This icon is available in the CAM module as well.

Ridge

This function converts a single-line drawing into a closed contour. This offset curve can be totally parameterized.

Select the contour to be converted and click on the icon **Ridge** . The following dialog box is displayed:



Keep current curves: the original curve is kept.
Dimensions and kind of corners: three types of angles are available: sharp, chamfered and rounded.
Ridge radius: enter the value that defines the offset.
Start and End of ridge: two geometries are available: flat and rounded.

Example

Single-line arrow *Arrow with contours* *Zoom*
(original curve kept - beginning and end of ridge rounded)

DELETING THE LAST OPERATIONS

Undo function

Undo
Cancel the last action (by default, up to the last 10).

Quick tip: **Ctrl + Z**

Redo function

Redo
Redo / cancel the last undo (by default, up to the last 10).

Remark

In the Edit **menu, you can find a reminder of the last operation executed (that could be canceled with** Undo**) and of the last operation canceled (that could be redone with** Redo**).**

Parameters of the Undo memory

It is possible to parameter the number of cancellation of modifications (Undo) done during the composition.
In the menu **Edit → Options...**, enter the desired number in the **Undo level** field.

Erasing the Undo memory

If you computer is limited in RAM memory, we advise you to erase the content of the Undo memory regularly by selecting the **Clear undo stack** function in the **Edit** menu. The last modifications done on the composition and saved in the memory, will be erased and it will not be possible to cancel or to redo them again.

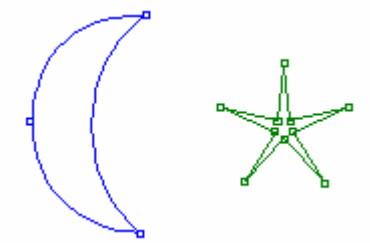
COMBINING AND UNCOMBINING OBJECTS

Combining

This function is useful to combine several objects into the same curve, even if there is no link between them.

Therefore when clicking on an object belonging to the group, all the objects of the group will be selected.

After combining several objects, you can work simultaneously on all the contours of the objects under the shaping mode.



Combine
The selected objects are converted into curves and become a whole object.

Uncombining

Uncombine
Each object becomes independent of the other.

Remark

Only elements of the same nature can be combined: curve/curve, text/text...

It is therefore not possible to combine a text with a curve (rectangle, circle, etc.) for instance.
To do it, you need to convert first the text into a curve.

SHAPING MODE

Shaping

This mode (also called modelling mode) is useful to modify the shape of an object or of a character font – when converted into curves – working directly on the points of construction along the contour.

Converting into curves

This function is used especially with texts. It is useful to create mirror effects, to retouch letters under the shaping mode, etc.

Convert into curves

Select the text with the selection tool and click on this icon.

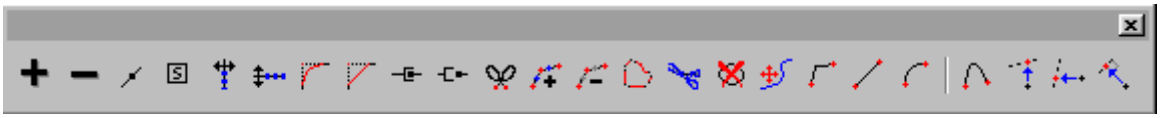
Shaping tool

Shaping tool

When clicking on the shaping tool, the outline displays the following:

- points located on the curve (indicated by little squares): □
- control points of the curve (indicated by little circles): ○

The new toolbox for shaping has different options permitting to add or delete a point, to connect or disconnect a contour, to modify the shape of curves, and to change the starting point.



Remark

You can put any selected tool back down by hitting the space bar.

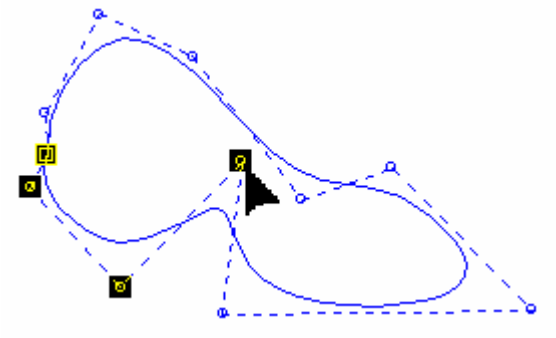
Selecting points

Two methods can be used to select several points under the shaping mode.

Selecting several points one by one

To select several points, click on each point while pressing the **Ctrl** key.
The selected points appear in a black square.

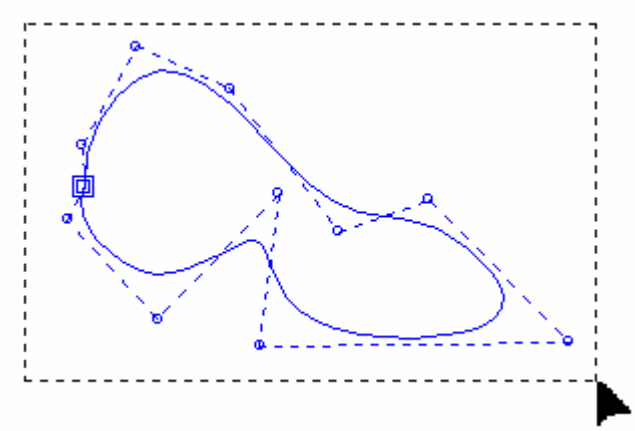
Example



Selecting several points at once

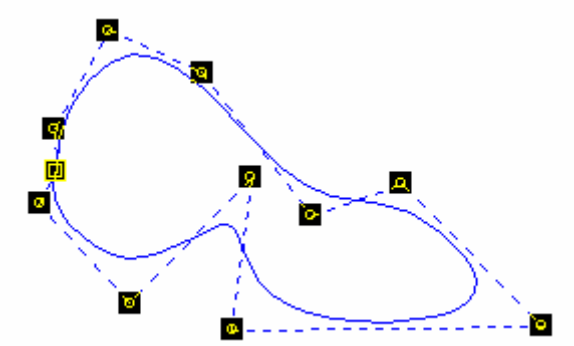
To select several points at once, click on the left button of the mouse and keep it pressed. Drag the mouse until all the desired points are selected in the selection window (represented by a dotted line on the screen).

Example



The selected points appear in a black square.

Example



Adding and deleting points

Add a point

Click on the shape where you want to add a control point, (a point appears on the contour) and then click on this icon, or press the "**+**" key on your keyboard.

Remark

When you add a point, it is always positioned on the curve.

Delete a point

Select the points to be deleted and click on this icon, or press the "**-**" or **Del** key on your keyboard.

Remark

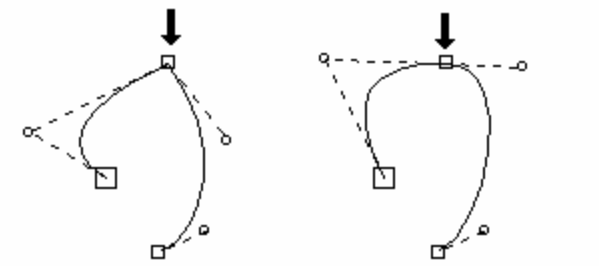
It is impossible to delete the starting point of a contour (double square).
To delete it, it is necessary to define another starting point.

Altering the control points

Continuous control point
This function changes the selected point into a tangent point of the curve.

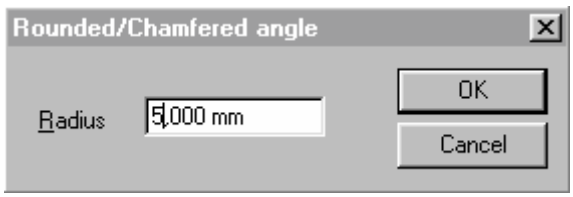
The position of the selected point (point on curve) is not modified. Only the 2 points located outside of the curve are changed to become the tangent point.

Example



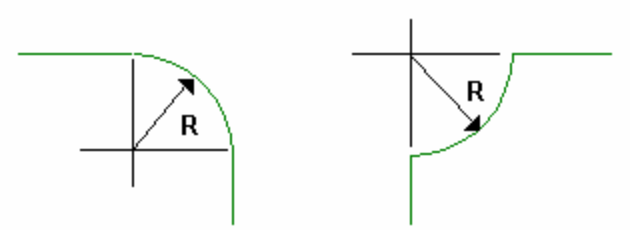
angle point *continuous control point*

Rounded angle
Select the points on the angle you want to modify and click on the icon.
Enter the value of the radius of the angle in the dialog box **Rounded/Chamfered angle**.



If the value of the radius is positive, the angle will be created outside the contour.
If the value of the radius is negative, the angle will be created inside the contour.

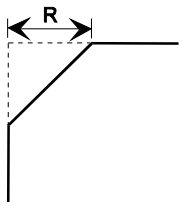
Example



Positive radius *Negative radius*

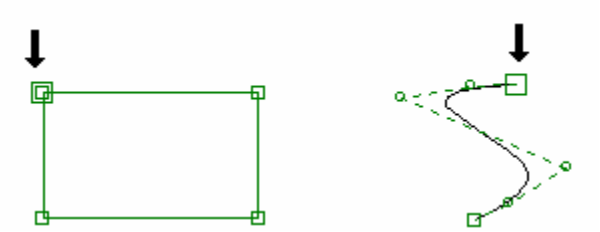
Chamfered angle

Select the point on the angle you want to modify and click on the icon.
Enter the value of the chamfer in the dialog box **Rounded/Chamfered angle**.



Starting point

The starting point on a closed contour is indicated by a double square.
The starting point on an open contour is indicated by a big square.
Example



Closed contour *Opened contour*

Modifying the starting point

Closed contour
Select the new starting point, which must be on the curve, and click on the icon.

Open contour
To change the starting point of an open contour (indicated by a big square), you need to select first the **Reverse direction** icon .

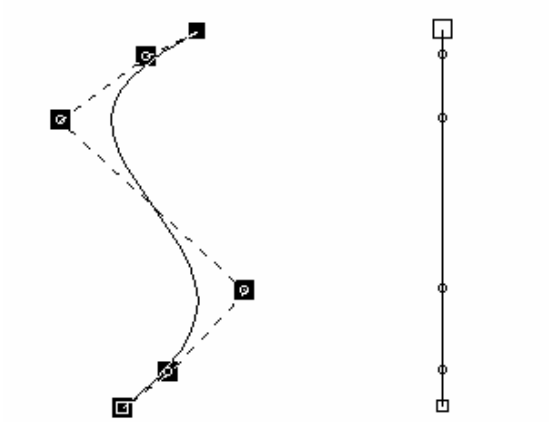
Remark
It is impossible to delete the starting point of a closed contour.
To delete it, it is necessary to define another starting point.

Aligning points vertically or horizontally

Aligning the points vertically

While pressing the **Ctrl** key, select the points to be vertically aligned and click on the icon.
The **X** value of the first selected point will determine the alignment.

Example

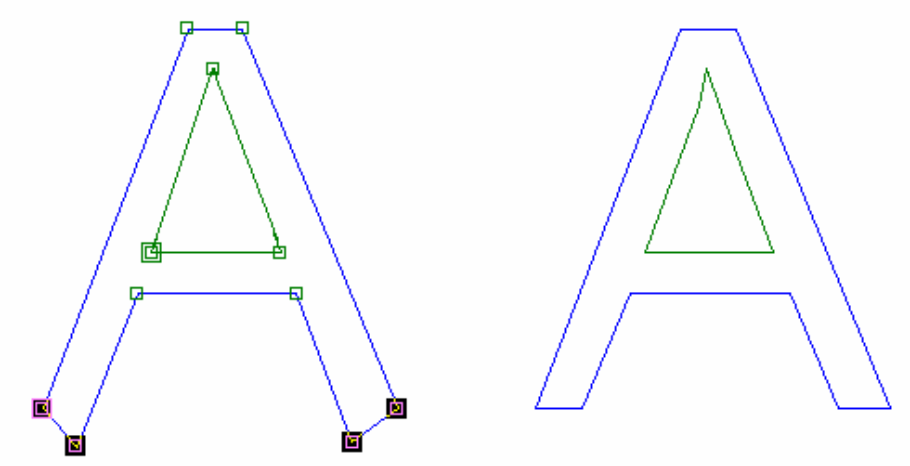


Selected points *Aligned points*

Aligning the points horizontally

While pressing the **Ctrl** key, select the points to be horizontally aligned and click on the icon.
The **Y** value of the first selected point will determine the alignment.

Example



Selected points *Aligned points*

Disconnecting and connecting a contour

Disconnect
Add a point where you want to disconnect the curve or select an existing point on the curve and click on this icon.

Connect
Select the two end points and then click on this icon to join the contours.

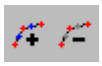
Remark
To connect points issuing from different shapes, you first have to select them, and then combine them.

Auto-connect
Select the points to be connected and click on the icon.
A dialog box is displayed by clicking simultaneously on the **Connection** icon and **Ctrl** key. Then, you can parameterize the distance allowing the connection and/or the merging of the points, as well as the connection mode (addition of a segment between points, superposition of points before connecting them).

The **Separately** checkbox enables to link points without any merging between them. Therefore, all entities will be kept.

Remark
We advise you to use this function for the DXF **with open contours issued from an** Autocad **software, for instance.**

Refine and simplify a curve

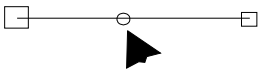
 Select a curve to which you want to add points and click on the **Refine curve** icon**:** a new point is created between each pair of existing points. Conversely, the **Curve simplification** icon reduces the number of control points according to a customized Tolerance.

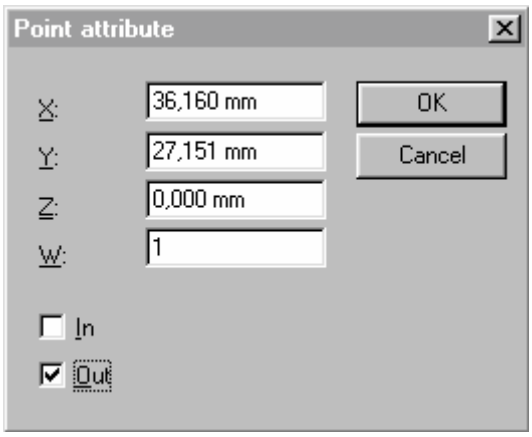
Explode a curve

Select a curve you wish to split up into simple elements and click on the **Explode a curve** icon**:** the curve is broken up at each control point and each portion can be manipulated individually.

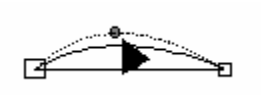
Shaping straight lines and curves

To change a straight line into a curve
Add a point on the straight line and double-click on this point.

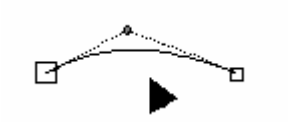




The dialog box **Point attribute** appears indicating the position of the point you have just created.



Select the option **Out of curve**.
The point becomes a control point of the curve.



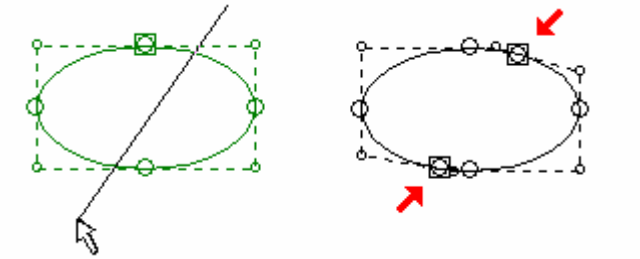
To modify the curvature of the segment with the cursor drag the point where desired.

Cut

This tool allows to cut one or several combined contours.

Click on the **Cut** icon and drag the mouse to draw a segment passing on the contour.
The CAD module automatically creates two cutting points at each intersection of the segment and the contour.

Example



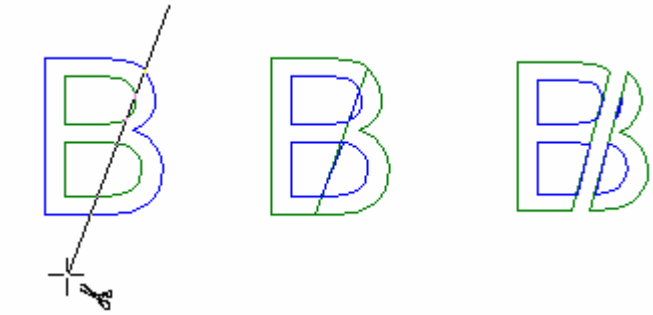
Shift + Cut

Associating the **Cut** function and **Shift** allows to cut an element and to close automatically the created contours.

Click on the **Cut** icon. Press then the **Shift** key and keep it pressed meanwhile you drag the mouse to draw a segment passing on the contour of the object you want to cut.

The CAD module automatically cuts the contour at the points of intersection and creates closed contours.

Example



Remark

This function can only be used on closed contours.

Deleting a contour

You can delete a contour independently of other contours of a drawing. Using this function, you don't need to uncombine the contours to delete one of them.
Click on the **Deleting a contour** icon, and select then the contour you want to delete.

Moving a contour

You can move a contour without having to uncombine it from others.
Click on the **Moving a contour** icon. Select then the contour you want to move, and drag the mouse.

Linearization tools

The following tools are useful to modify a part of a vectorized contour.

Linearize

Click on the **Linearize** icon. Then put the cursor on the contour and click on the segment starting point.
Put the cursor on the segment arrival point and click to end this operation. The line between both points is automatically transformed into a straight line.

Linearize in arc of circle

Click on the **Linearize in arc of circle** icon. Place the cursor on the contour where you want the arc starting point to be. Put the cursor on the arc arrival point and click on it to define the conversion area. Drag then the mouse to define the arc. When the desired arc is obtained, click a last time to end the operation.
The area between both points is automatically transformed into an arc of circle.

Linearize in curve

Click on the **Linearize in curve** icon. Place the cursor on the contour where you want the curve starting point to be. Put the cursor on the curve arrival point and click on it to define the conversion area. Drag then the mouse to define the curve shape. When the desired curve is obtained, click a last time to end the operation.
The area between both points is automatically transformed into a curve.

Linearize in angle

Click on the **Linearize in angle** icon. Place the cursor on the contour where you want the curve starting point to be. Put then the cursor on the arrival point and click on it to define the conversion area. Drag then to define the angle of the curve and click a last time once you have got the desired angle.
The area between both points is automatically transformed into a broken line.

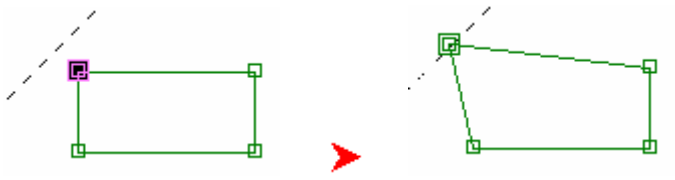
Projecting points

This function allows to project one or several points on a defined guide.

Normal projection

First select the point(s) you want to be projected.
Click on the **Normal projection** icon. Put the cursor on the guide where you want the point(s) to be projected and click on the left button of the mouse.
The point is orthogonally projected on the guide.

Example

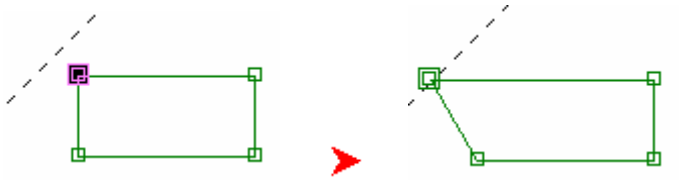


Original drawing *Result*

Horizontal projection

First select the point(s) you want to be projected.
Click on the **Horizontal projection** icon. Put the cursor on the guide where you want the point(s) to be projected and click on the left button of the mouse.
The point is horizontally projected on the guide.

Example

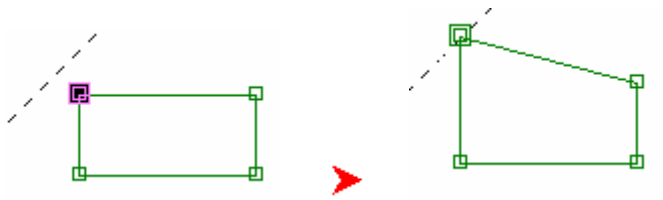


Original drawing *Result*

Vertical projection

First select the point(s) you want to be projected.
Click on the **Vertical projection** icon. Put the cursor on the guide where you want the point(s) to be projected and click on the left button of the mouse.
The point is vertically projected on the guide.

Example



Original drawing *Result*

REGISTERING MARKERS

Positioning registering markers

Registering markers

The registering markers which allows spotting in Vinyl cutting are available in the symbol library under the directory **Markers**:

hirondel1.syb

hirondel2.syb

SURFACE'S TOOLS

Displaying the surface's toolbar

You can get access to the **Surface's toolbar** from the CAD module left toolbar by clicking on the **Surface's tools** icon .

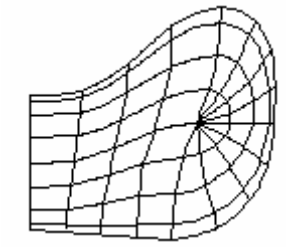
The corresponding toolbar is displayed on the screen (See following sections **Handling surfaces** and **Projection and mapping** to know more about the use of these tools).

HANDLING SURFACES

Import of the surface described in IGES

Make the drawing in the CAD module. Then create the tool path(s) with **CAD** module (the blue screen). Each tool path is saved with its own name. The first will be named **Name of drawing.000**, the second **Name of drawing.001**, etc.
For instance, if your drawing is called "Type3" and is in the ".vnd" path (Type3.vnd), the first tool path will be Type3.000.

Importing an IGES surface
Click on the shown icon or select **Import** from the **File** menu. Choose the format **IGES file [*.IGS]**.



Remark

Check to make sure you are only importing a non-trimmed or a non reduced surface (without rotation, nor scale) and without holes or overlapping areas. The matrices are not read.
To import surfaces written in IGES format, the software only imports 128 entities.

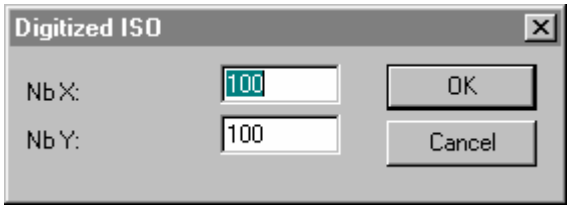
Modifying surfaces
To modify or to clean the surfaces if needed, you must first uncombine them.
Do not forget to recombine them afterwards.

Displaying surface step

The surface step corresponds to the number of lines that describes the surface.
The default value is 5 isometrics. To modify the display parameter, go to the menu **Edit → Options…** and put the new value in the field **Surface step**.

Importing a digitized surface

Importation
Click on the shown icon or select **Import** from the **File** menu and choose the **ISO digitized [*.ISO]** format.



Nb X and Nb Y
Number of points on the X and Y axis following the step of digitalization.

Sampling
To find the number of points in X and Y of a regular digitalization, import the surface in simple ISO.

Then go in mode point with the curve shaping tool. Select the points on the X axis of the surface. The status bar found on the bottom of the screen gives you the number of points on the X axis. Repeat the same operation for the Y axis.

Remark
Avoid selecting the starting point which has 2 points.

Import the surface again but this time selecting it in the **digitized ISO** format. Then enter the corresponding number of points in X and Y in the dialog box.

Creating simple 3D surfaces

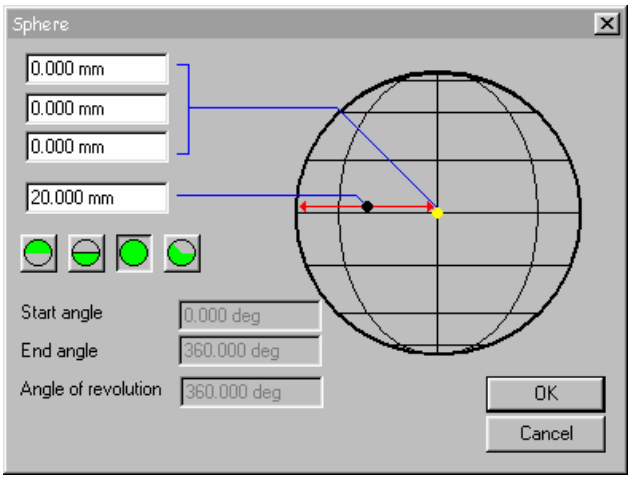
You can create simple 3D surfaces (spheres, cylinders, cones, ellipsoids, cubes and tores) on which you will brace or map.
After their creation, simple and complex surfaces can get merged, subtracted, extended, etc. You can also convert any kind of TypeArt object into a surface entity giving you the possibility to export with the IGES format.
The corresponding tools are located in the **Surfaces** toolbar. These tools replace in a more visual way the **3D Bracing** functions.

Creating a sphere

In the **Surfaces** toolbar, click on the **Sphere** icon . Position the cursor on the material where you want the sphere center to be and click the mouse left button. Drag the cursor until the sphere has the desired dimensions. Click another time the mouse left button to validate the created shape.

Remark
By default, you will only create the upper half of the sphere. Hit the Shift **key (while you drag your cursor) to create a whole sphere or the** Ctrl **key (meanwhile you drag the cursor) to create the lower half of the sphere.**

Hit the **F2** key (once you clicked on the **Sphere** icon) to create the sphere numerically. The dialog box **Sphere** appears on the screen.



By default, a whole sphere is created. Click on the desired icon of the dialog box to generate the upper or the lower part of the dialog box or any part of it by stating its start, end and revolution angles.

Enter the values corresponding to the sphere radius (20 mm in the example above) and to its center position (0 mm). Click then on **OK** to validate.

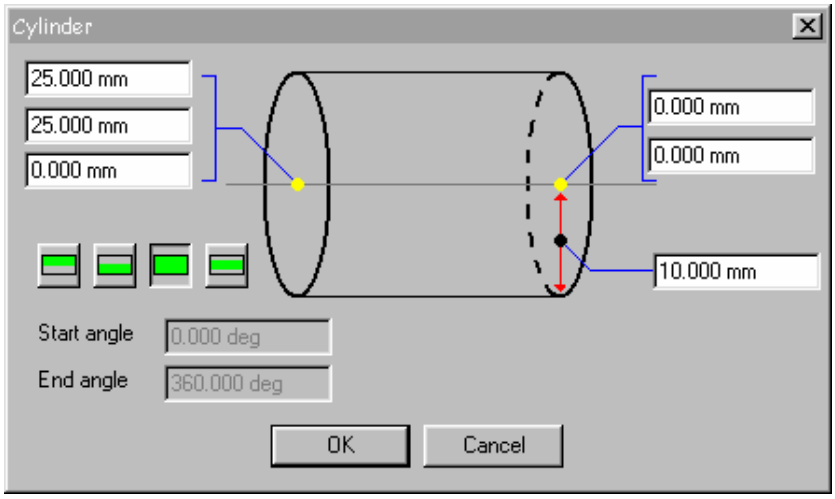
Creating a cylinder

In the **Surfaces** toolbar, click on the **Cylinder** icon . Position the cursor on the material where you want the center of the cylinder's first side to be and click on the mouse left button. Drag the cursor until you reach the desired diameter and click once again. Drag now the cursor until the cylinder has the desired direction and dimensions. During this step, the cylinder is pivoting around the center of the cylinder's first side. Click a last time to validate the created shape.

Remark

By default, you will only create the upper half of the cylinder. Hit the Shift **key (while you drag your cursor) to create a whole cylinder or the** Ctrl **key (meanwhile you drag the cursor) to create the lower half of the cylinder.**

Hit the **F2** key (once you clicked on the **Cylinder** icon) to create the cylinder numerically. The dialog box **Cylinder** appears on the screen.



By default, a whole cylinder is created. Click on the desired icon of the dialog box to generate the upper or the lower part of the cylinder, or any part of it by stating its start and end angles.

Enter the values corresponding to the coordinates of the center of the cylinder's first side (25 mm in the example above), of the second side's center (0 mm in the example) and of the radius (10 mm in the example). Validate with **OK**.

Creating a cone

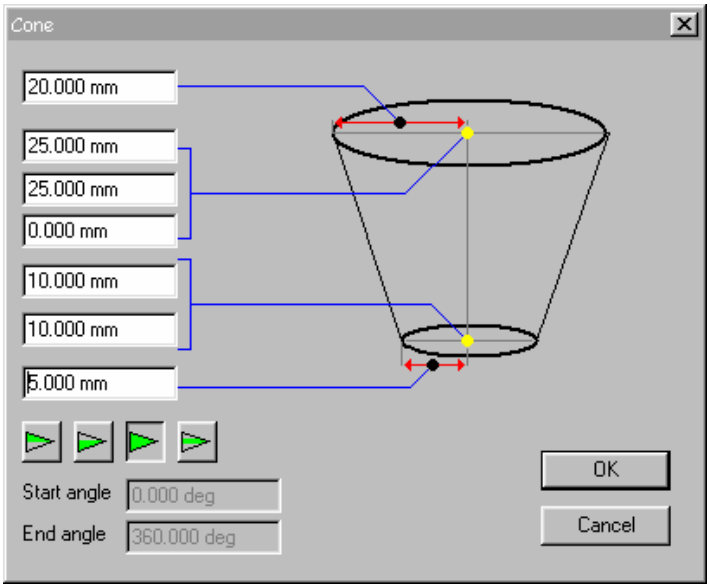
In the **Mapping** toolbar, click on the **Cone** icon . Position the cursor on the material where you want the center of the cone's finest end to be and click the mouse left button. Drag the cursor until the diameter of this end is reached. Validate by clicking once again the mouse left button. Drag now the cursor until the cone has the desired direction and dimensions. During this step, the cone is pivoting around the center of its finest end. Click a last time the mouse left button to validate the created shape.

Remark

By default, you will only create the upper half of the cone. Hit the Shift **key (while you drag your cursor) to create a whole cone or the** Ctrl **key (meanwhile you drag the cursor) to create the lower half of the cone.**

Hit the **F2** key (once you clicked on the **Cone** icon) to create the cone numerically.

The dialog box **Cone** appears on the screen.



By default, a whole cone is created. Click on the desired icon of the dialog box to generate the upper or the lower part of the cone or any part of it by defining its start and end angles.
Enter the numerical values in the appropriate fields:

- Radius of the finest end (5 mm in the example shown above).
- Coordinates of the finest end's center (twice 10 mm in the example).
- Coordinates of the widest end's center (twice 25 mm in the example).
- Radius of the widest end (20 mm in the example).

Click then on **OK** to validate.

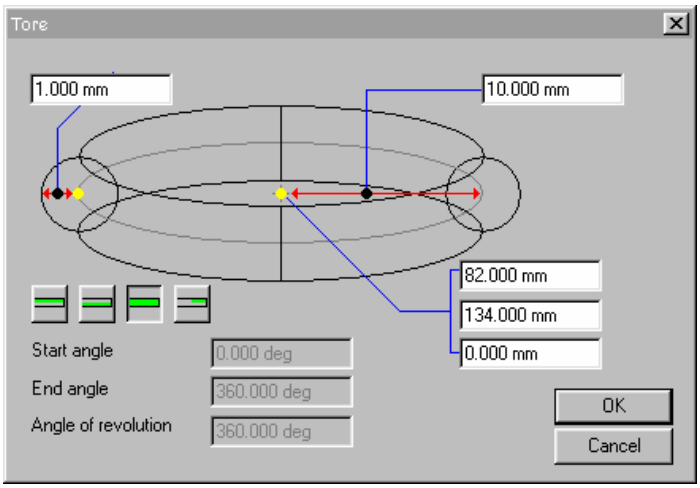
Creating a torus

In the **Surfaces** toolbar, click on the **Torus** icon . Position the cursor on the material where you want the torus center to be and click on the mouse left button. Drag the cursor to define the radius of the torus cross section (it corresponds to the created circle on the screen). Click the mouse left button to validate. Drag now the cursor to define the radius of the torus until the torus reaches the desired dimensions. Click a last time to validate the created shape.

Remark

By default, you will only create the upper half of the torus. Hit the Shift **key (while you drag your cursor) to create a whole torus or the** Ctrl **key (meanwhile you drag the cursor) to create the lower half of the torus.**

Hit the **F2** key (once you clicked the **Torus** icon) to create the torus numerically. The dialog box **Torus** appears on the screen.



By default, the whole torus is created. Click on the desired icon of the dialog box to generate the upper and lower part of the torus or any part of it by stating its start, end and revolution angles.

Enter then the desired values in the corresponding fields:

- Radius of the cross section (1 mm in the example shown above).
- Radius of the torus (10 mm in the example).
- Coordinates of the center of the radius (82 and 134 mm in the example).

Click then on **OK** to validate.

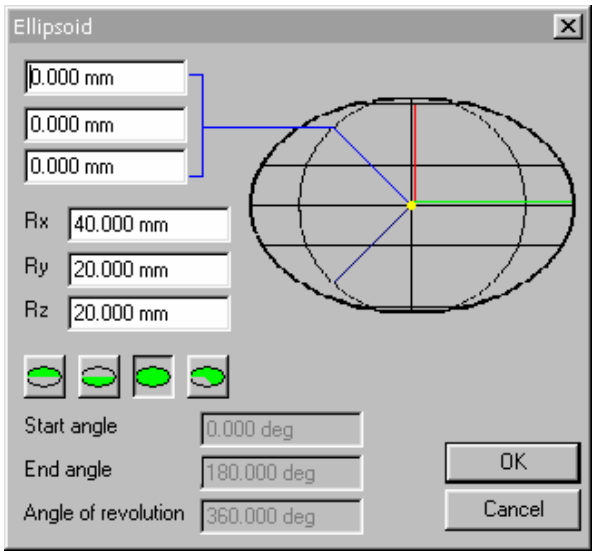
Creating an ellipsoid patch

In the **Surfaces** toolbar, click on the **Ellipsoid patch** icon . Position the cursor on the material where you want the ellipsoid patch center to be and click on the mouse left button. Drag now the cursor until the ellipsoid patch reaches the desired dimensions. Click a last time the mouse left button to validate the created shape.

Remark

By default, you will only create the upper half of the ellipse. Hit the Shift **key (while you drag the cursor) to create a whole ellipse or the** Ctrl **key (while you drag the cursor) to create the lower half of the ellipse.**

Hit the **F2** key (once you clicked on the **Ellipsoid patch** icon) to create the ellipsoid patch numerically. The corresponding dialog box is displayed on the screen.



By default, a whole ellipsoid patch is created. Click on the desired icon of the dialog box to generate the upper or the lower part of the ellipsoid patch or any part of it by stating its start, end and revolution angles.

Enter the values corresponding to the radii (40 and 20 mm in the example above) and the position of the shape center (0 mm in the example above).

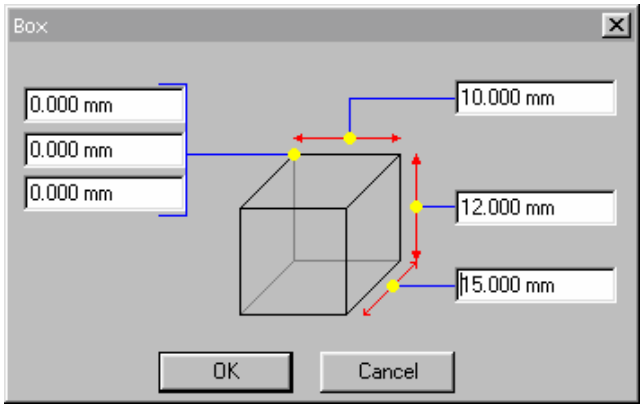
Click then on **OK** to validate.

Creating a box

In the **Surfaces** toolbar, click on the **Box** icon . Position the cursor on the material where you want the box center to be and click the mouse left button. Drag the cursor until the box reaches the desired dimensions. Click a last time the mouse left button to validate the created shape.

Hit the **F2** key (once you clicked on the **Box** icon) to create the box numerically.

The **Box** dialog box is displayed on the screen.



Enter the values defining the position of the box (left numerical fields) as well as the ones defining its dimensions (right numerical fields).

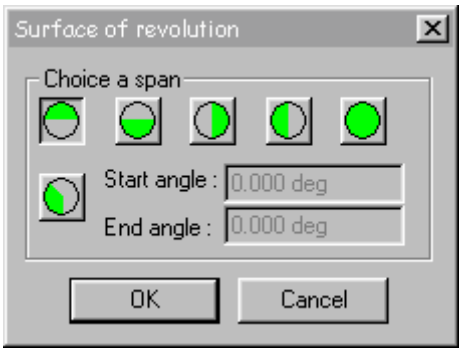
Click then on **OK** to validate.

Creating a surface of revolution

How to create the surface
With CAD module tools, draw the section and define the axis of rotation with two perfectly aligned markers.

Select the section first, then the two markers.

Click on the **Surface of revolution** icon . The corresponding dialog box is displayed on the screen.

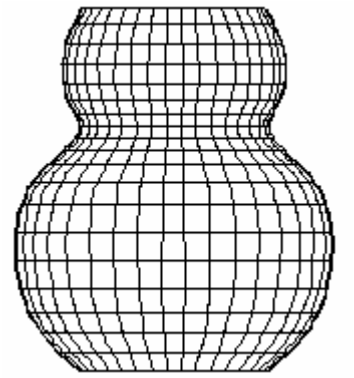


By default, the upper half of the surface of revolution will be generated. Click on the desired button of the dialog box to generate the lower, the right or the left half, or also the whole surface of revolution (this complete rotation represents 360°).

You can also make a partial rotation by clicking on the button of the dialog box: enter then the **Start angle** and the **End angle** of the surface.

Example

1/2 concave: 180° to 0°
1/2 convex: -180° to 0°



Modifying the display of the surface step
Default value: 5 isometrics. To modify this display parameter, select **Options…** in the **Edit** menu (or hit the **F10** key). The dialog box **Options** is displayed on the screen: click on the **Display** thumbnail. You can now modify this parameter by entering the desired value in the **Surface step** field.

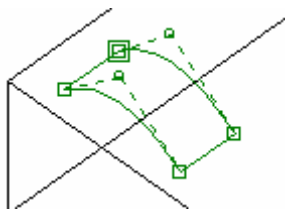
Creating a surface of Coons

Draw a closed contour in the CAD module. Permute to the **Point mode**. Select the point(s) which altitude must be modified. You can add points on segments to modify their altitude on accurate parts. Double-click on the points you want to modify. The dialog box **Point attribute** is displayed.

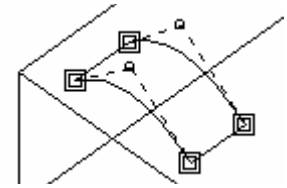
Commute then to the 3D ISO view to visualize the shape of your surface of Coons. Therefore, double-click on the icon

located in the status bar at the bottom of your screen. In the dialog box displayed on your screen, click on **3D ISO** and validate.

Example



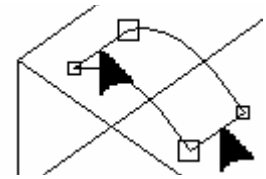
Disconnect then the points of the original contour by selecting them and clicking on the **Disconnect point** of the **Point mode** toolbar. Once all the points are disconnected, the contour becomes black. It proves it became an open contour.



Select the whole contour and click on the **Uncombine curves** icon in the toolbox. Each curve of the contour is now independent.

Contours direction
In order to create a surface of Coons and to make then a bracing operation in the best conditions, you must respect this requirement: opposite segments must have the same direction. Ensuring the contours direction has to be made under the point mode.

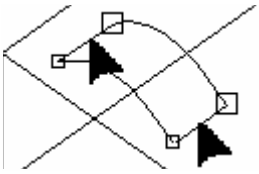
Example



Pointed contours don't have the same direction.

To invert a curve direction (this can only be made under the point mode), select the curve and click on the icon **Reverse contours**.

Example

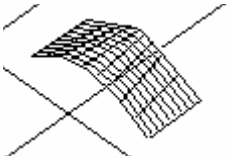


Pointed contours have now the same direction.

Selection order of the contours
This is also made under the point mode.
First select the upper contour and by pressing and holding down the **Ctrl** key select the opposite contour, then the left one and last the right one.

Surface of Coons
Once all contours are selected, click on the **Surface of Coons** icon.

Example



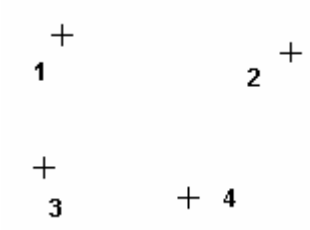
Your surface of Coons is created

Creating a bilinear surface

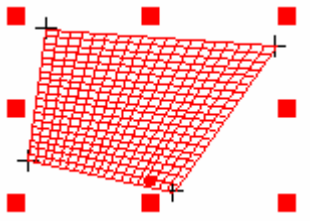
This kind of surface has to be drawn using 4 markers. After selection, these markers are bounded by a bilinear surface if you click on the **Bilinear surface** icon .

The surface shape will be defined by the order of selection of the markers.

Clockwise selection of the markers
This is also the default selection mode in case you make use of a selection window. Otherwise, click on each marker to be selected while keeping pushed the **Ctrl** key as shown in the example below.

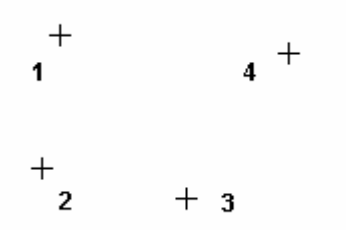


Order of selection

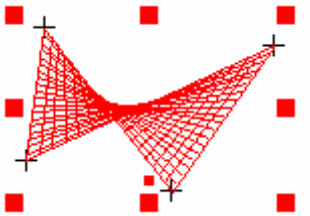


Obtained surface

Anti-clockwise selection
Click on each marker to be selected keeping pushed the **Ctrl** key as shown in the example below.

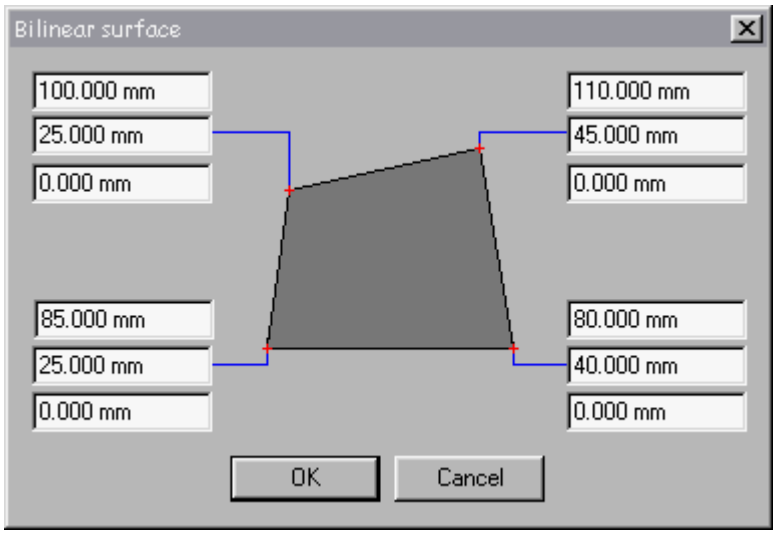


Order of selection

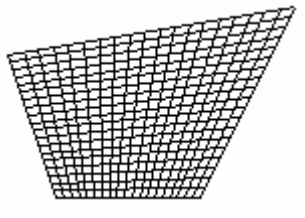


Obtained surface

You can also directly use the sketching tool to draw your surface freely. Click therefore on the **Sketching tool** icon . Drag your cursor and click once each time you want to create an angle. Hit the **F2** key to enter accurate numerical values in the displayed dialog box for the creation of this surface.



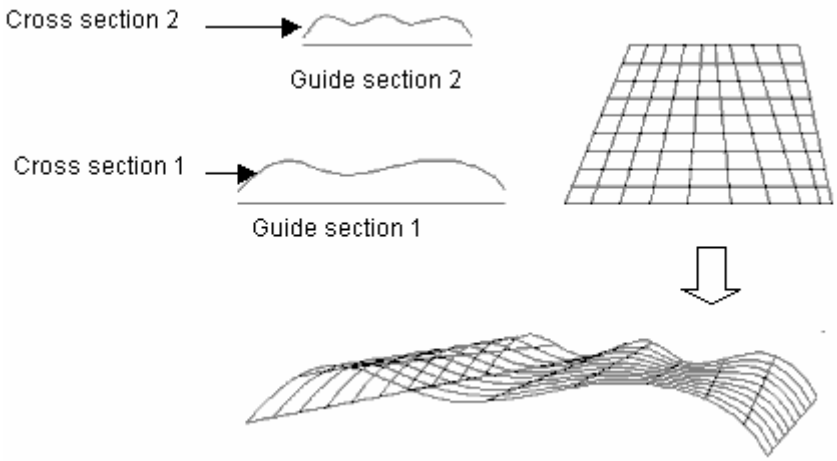
Enter the same values as in the example shown above to obtain the result shown below.



Creating a ruled surface

Ruled surfaces are defined by two cross sections and two guide sections. Each cross sections is associated with a guide section. The surface is then interpolated between the cross sections.

Example



Remark

The selection order of the different elements as well as the direction of the selected curves is of great importance. Generally, the direction of each cross section must remain the same (from left to right or from right to left but never both). This also applies to these guide sections.

Selection order

First select the first cross section and then the second cross section. Select then the first guide section and finally the second guide section.

Once you've selected all the elements, click on the **Ruled surface** icon . The whole selection turned into a ruled surface.

Creating a swept surface

A swept surface is defined by a path, a cross section and a guide section. The cross section is used all along the path.

Remark

The selection order of the different elements as well as the direction of the selected curves is of great importance. Generally, the direction of each cross section must remain the same (from left to right or from right to left but never both). This also applies to the guide sections. In the case of a swept surface, cross and guide sections can stand side by side because of the cross section remains the same all along the path.

Selection order
First select the path, then the cross section and finally the guide section.

Once you've selected the required elements, click on the **Swept surface** icon . The swept surface is created.

Creating skinned surfaces

Skinned surfaces are defined by a certain number of cross sections (3 and more) associated with as many guide sections, with one or two paths optionally. Consequently, such surfaces can be very complex.

Remark

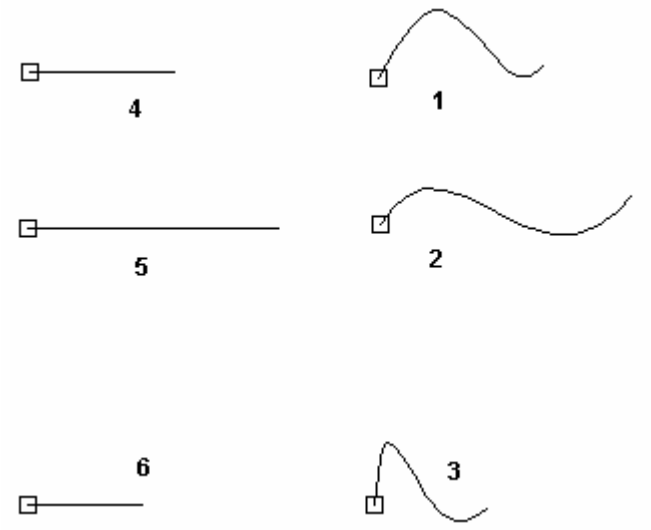
The selection order of the different elements as well as the direction of the selected curves is of great importance. Generally, the direction of cross and guide sections must remain the same (from left to right or from right to left but never both).

The way the guide sections are positioned is drastically fundamental. Actually, it defines the modification of the surface section. On the contrary, the way the cross sections are positioned is not important: the only selection order will be taken into account.

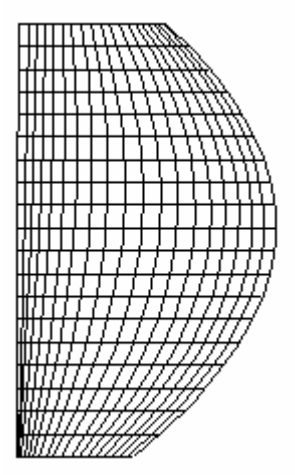
1. Simple skinned surfaces

Such surfaces are similar to ruled surfaces. The only difference is they can be made by using more than two cross sections and two guide sections.
First select the cross sections in their order of succession and finally the guide sections in the same order. Click now on the **Skinned surface** icon .

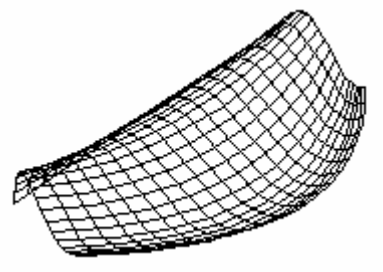
Example



Guide sections *Cross sections*



Result with a 2D view

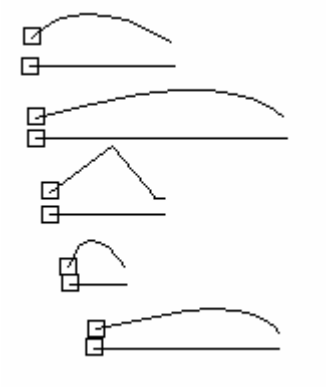


Result with a 3D Iso view

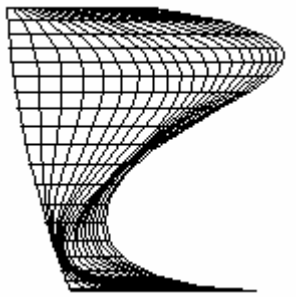
The result would have been exactly the same if the cross sections would have been positioned otherwise (but the selection order must remain the same).

The length of the guide section is very important because it determines the surface dimensions. By default, the cross sections are distended or compressed to occupy the whole length of the corresponding guide section. Nevertheless, if you hit the **Ctrl** key meanwhile you click on the **Skinned surface** icon , the length of the cross sections will prevail over the length of the guide sections.

Example with five cross sections



Cross and guide sections



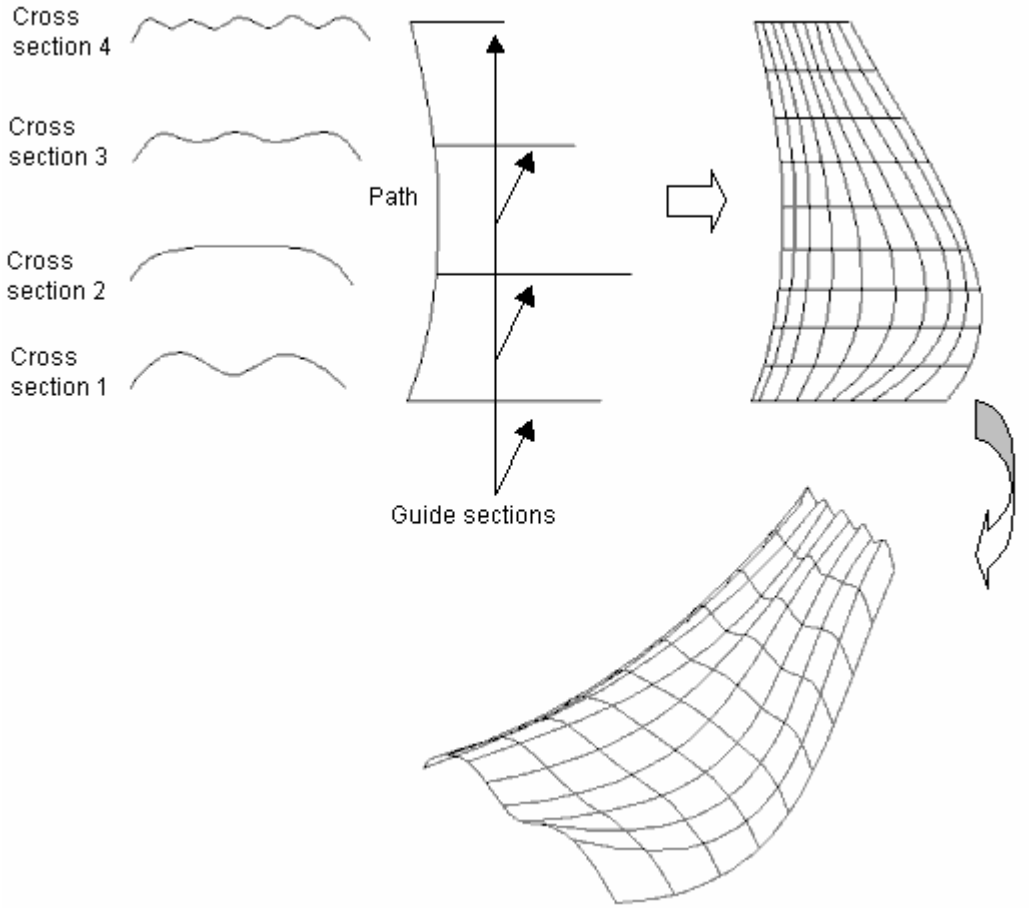
Result in 2D

Result in 3D Iso

2. Skinned surface with one rail

The only difference with the simple skinned surface is the use of a path applied to the side of the surface. The starting point of each guide section must be positioned along the path. For this positioning operation, use the snap options **Control point** or **Along the contour**. You can access these options by hitting the **F3** key or selecting **Snap mode…** in the **Edit** menu. In the displayed dialog box, activate (control point) or (along the contour) according to your needs.

Example

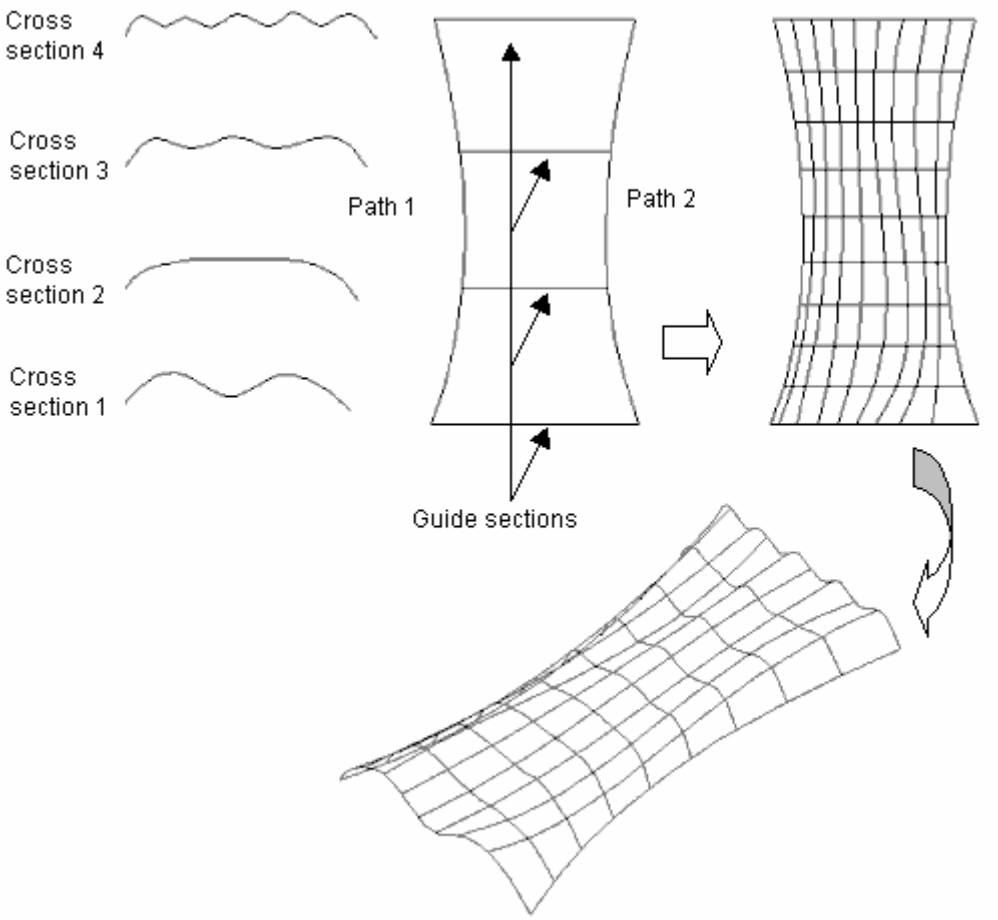


First select the path, then the cross sections and finally the corresponding guide sections. Click then on the **Skinned surface with one path** . Your surface is created.

3. Skinned surface with two paths

The only difference with the simple skinned surface is the use of two paths applied to the side of the surface. Starting points as well as ending points have to be positioned on the paths very accurately. For this positioning operation, use the snap options **Control point** or **Along the contour**. You can access these options by hitting the **F3** key or selecting **Snap mode…** in the **Edit** menu.

Example



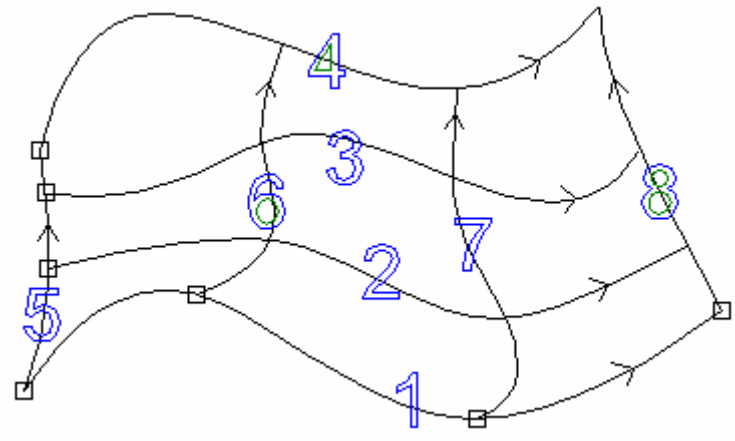
First select both paths, then the cross sections and finally the corresponding guide sections. Click then on the **Skinned surface with two paths** icon . Your surface is created.

Coons network surface

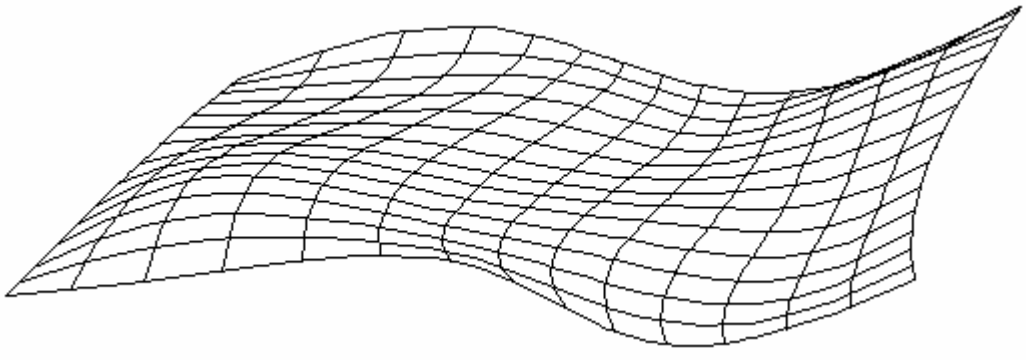
The Coons network surface is a complex surface, the structure principles of which are those of a Coons surface, i.e.:

- The curves have to have the same direction (vertical or horizontal),
- The curves must be linked together by using in the **Snap up** mode the **Control points** or **Along the contour** (**F3** key),
- The selection order is similar to the one of a complex envelope:
  Select first all horizontal curves, from the one which starting point is linked to the starting point of the first vertical curve (See figure: point at the left bottom),
- Click on the icon .

Examples



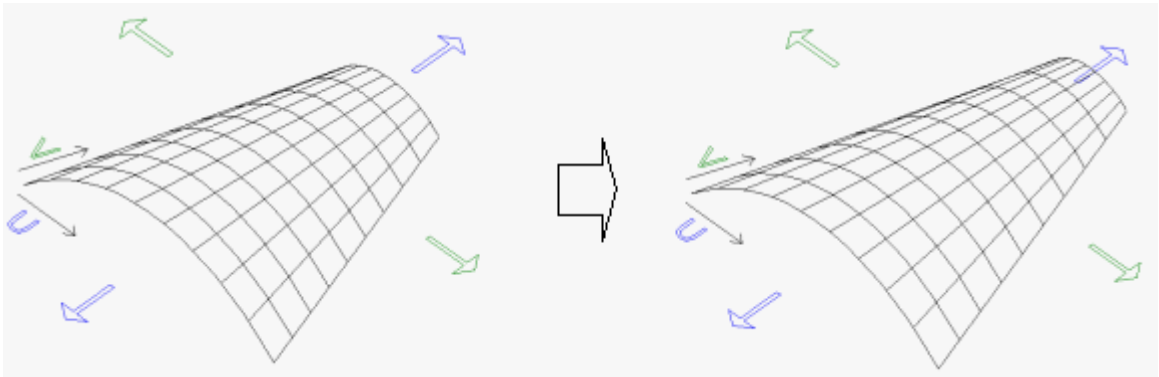
Curves for a network surface with curve direction and their selection order



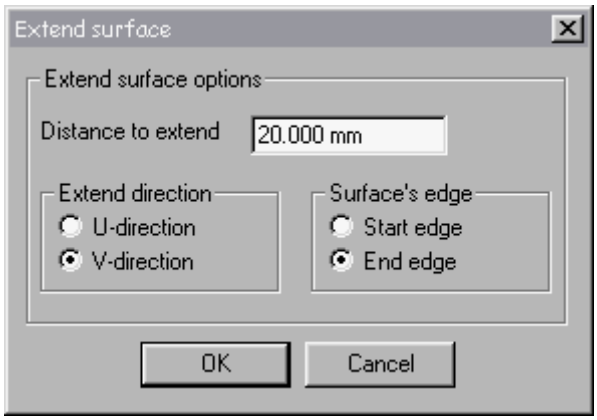
Result: Coons network surface

Extending a surface

This feature allows you to extend an untrimmed surface on one if its sides.



Select the surface to be extended and click on the **Extend surface** icon . The corresponding dialog box is displayed on the screen.



In the example shown above:

- **U-direction** corresponds to the **X-axis**.
- **V-direction** corresponds to the **Y-axis**.
- **Start edge** corresponds to the surface's left side for the U-direction and to the surface's bottom for the V-direction.
- **End edge** corresponds to the surface's right side for the U-direction and to the surface's top for the V-direction.

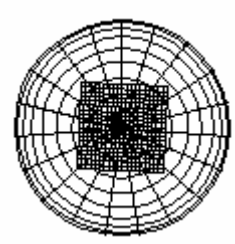
Always in our example, the surface is extended of 20 mm from the top of the surface.

Merging surfaces

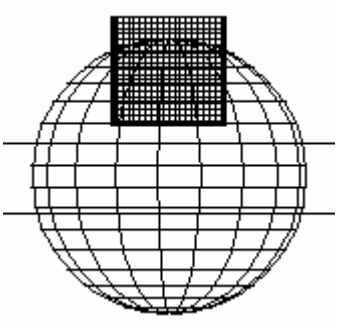
The CAD module puts a tool able to merge surfaces to obtain a single surface.

Select the surfaces to be merged (keep pushed the **Ctrl** key meanwhile you click on each surfaces to be selected) and click on the **Merge surfaces** icon .
If the surfaces have intersection areas, you can separate significant elements and keep interesting parts by uncombining the merged surface.

In the next example, we merge a sphere and a cube positioned as follows:

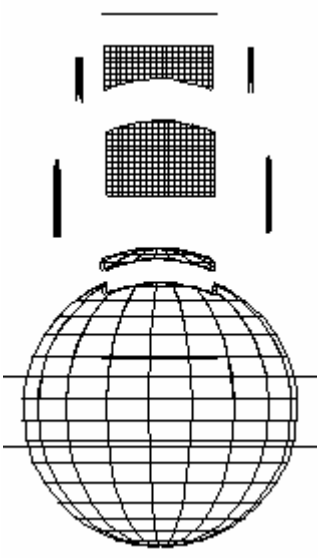


2D XY view

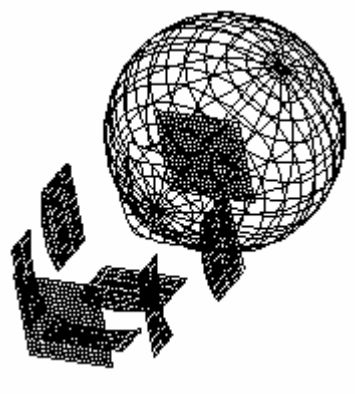


2D YZ view

You must now uncombine the merged surface using the tool. You can now remove certain elements for instance to scoop out the sphere using the cube shape.



2D YZ view

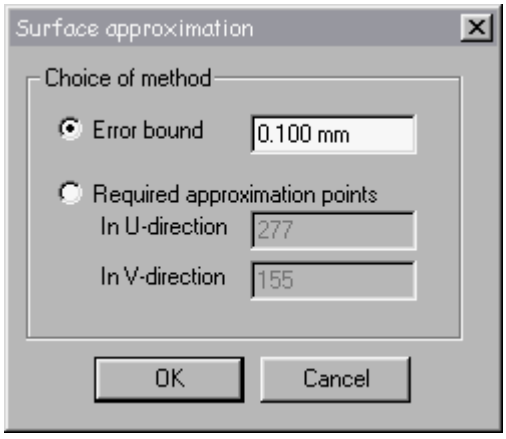


2D view…

Approximation of a TypeArt-object

This feature allows you to convert a TypeArt-object into a surface so that you can export it under the **IGES** format.

Select the TypeArt-object to be converted into surface and click on the **TypeArt approximation** icon . The corresponding dialog box appears on the screen.



You can choose among two methods to execute the TypeArt-object conversion.

- **Error bound**

This method consists in approximating between the created surface and the resulting surface.

Remark

The calculation for this method can very last very long!!!

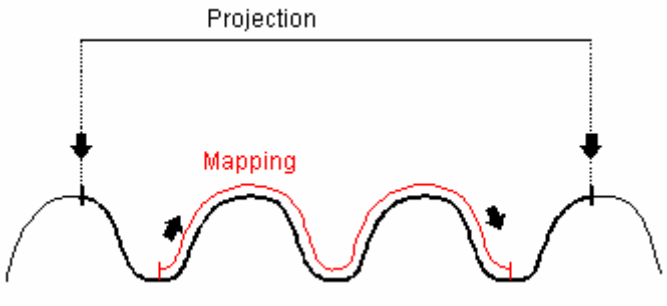
- **Required approximation points**

This method consists in defining the number of points for the U-direction (that corresponds to the X-axis) and for the V-direction (that corresponds to the Y-axis) as you would do by the creation of any TypeArt-object.

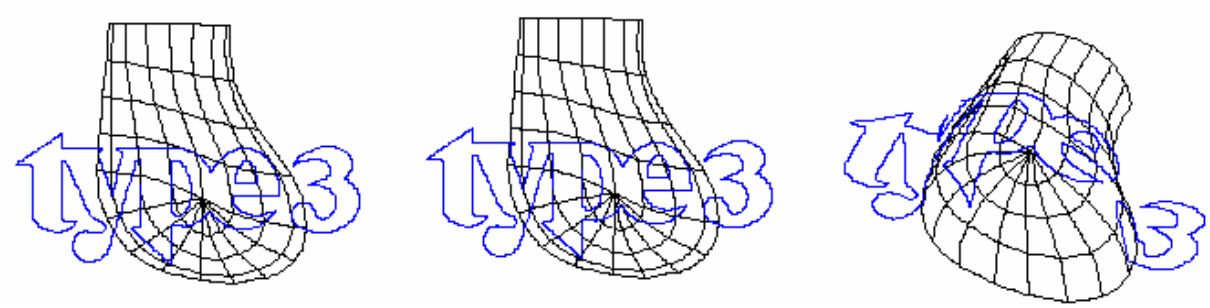
This approximation can not be greater than the TypeArt's resolution.

PROJECTION AND MAPPING

Principle for projection and mapping



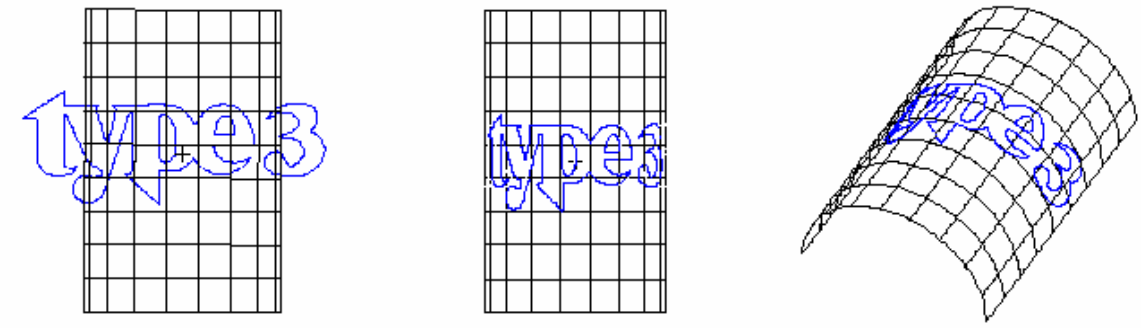
Projection
This function entails projecting along the Z axis. X and Y coordinates are not modified, so the object to be projected can go outside the borders of the surface.



Before projection *After: View in XY plane* *3D view*

If watched from the Z axis, the pattern is not distorted.

Mapping
The drawing is wrapped around the surface from a defined point, like a sheet of paper around a solid surface.



Before mapping *After: View in XY plane* *3D view*

Methods and tools for projection and mapping

Several methods are available for projecting or mapping a geometry on a shape: you will choose one, taking into account your type of design, the complexity of the shape, the number of dimensions of your toolpath, and your type of machine.

1. Parallel projection (for machining on 3 axes machines)

A – In the CAM module: When you select the surface and the pattern to project, an extra tab **Projection** appears defining your toolpath. See the section **Projection Tab** in the overview about toolpaths. It is the best method for projecting a toolpath aimed for a 3 axes machine.

B – In the CAD module: when you select the surface and the pattern to project, the **Projection** icon becomes active. See below **Parallel projection**.

2. Mapping using the post-processor (for machining on machines managing a 4th axis)

For cone and cylinder only: You have created your design in order to make a flat toolpath. The mapping is done by the post-processor that manages a 4th axis. In the **Machining** window, a **Cylinder** button appears, allowing to define the radius of the cylinder (2 radius for a cone).
This simple method is possible only on true cylindrical or conical shapes.

3. Mapping on complex shapes (for machining on machines managing a 4th axis)

For skewed shapes: You have created your design in order to make a flat toolpath. The toolpath itself will be arranged in the CAM or in the CAD module, with the projection/mapping tools.

Procedure 1: You have made your design and created on 2D the necessary toolpaths in the CAM module.

In the dialog box **Toolpaths list**, select the toolpath you want to project or map on your surface. Hit the right mouse button and choose the appropriate option at the bottom of the context menu. The dialog bow used are the same as those available in the CAD module (see below).

Use the projection/mapping tools (see below) to create the mapping, then go back in the CAM module.

Procedure 2: You have made your design and created on 2D the necessary toolpaths in the CAM module. Each path is registered under a name such as **Type3.vnd** ; the first one is labeled as **TYPE3.000**, the second **TYPE3.001**, etc.

In the dialog box **Toolpaths list**, select the toolpath you want to project or map on your surface. Hit the right mouse button and choose the option **Edit toolpath geometry** in the context menu. The display color of the toolpath changes. The toolpath is now accessible from the CAD module.

Use the projection/mapping tools (see below) to create the mapping, then go back in the CAM module.

Select the projected or mapped toolpath. Select then the toolpath to be replaced in the dialog box **Toolpath list**, and hit the mouse right button. Choose the option **Replace path** in the context menu

In the **Machining** window, a **Circular** button appears, allowing to define with two points the revolution axis for the mapping.

Projection and mapping tools of the CAD module

Directly applied to a pattern, these four tools are convenient for projection (method 1B), but not for mapping (unless you just want to see what the mapping will look like, without machining). They are also used for toolpaths (method 3).

Parallel projection

The pattern is laid on the surface and follows its shape, starting from a reference point.

Basic mapping
The pattern is mapped on the surface and follows its shape.

Cylindrical/Conical mapping /
The pattern is mapped on a cylinder / cone (respectively specified by one or two radius) then projected parallel to the surface.

Parallel projection

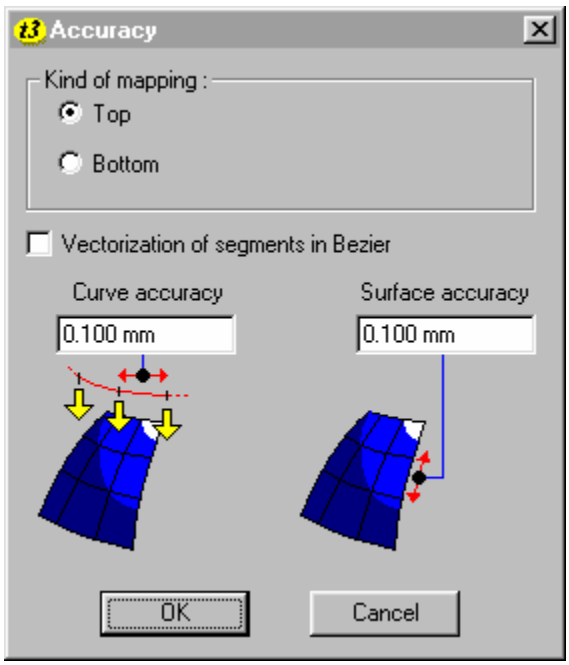
REMARK

We consider the projection are applied to toolpaths, though in some cases they are used directly on curves and patterns (see above Methods and tools for projection and mapping).

Projecting the toolpath
Place the toolpath(s) anywhere you would like on the surface (on the X, Y plane).

While holding down the **Ctrl** key, first select the toolpath and then the surface.
They both should become red and the icon **Parallel projection** becomes active.

Click on the icon **Parallel projection** . The dialog box **Accuracy** is displayed on the screen (the same dialog box can be accessed from the contextual menu of the CAM module).



Top
Activate this option to do a convex mapping.

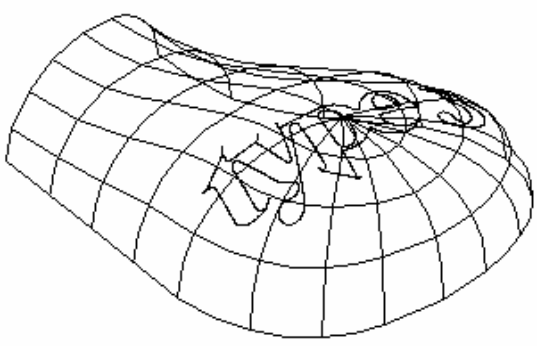
Bottom
Activate this option to do a concave mapping.

Vectorization of segments in Bezier
Activate this option to directly brace or map Bezier curves. The amount of points will be extremely decreased.

Curve accuracy / Surface accuracy
Define in both fields the accuracy of the mapping surface. The smaller the chosen value, the finer the surface mesh and the more accurate the projection will be. The computation will also be longer.

The calculation is done automatically after clicking on **OK**. A progression bar will indicate the status of the job.

To see the result, choose **3D ISO view** from the **View** menu.



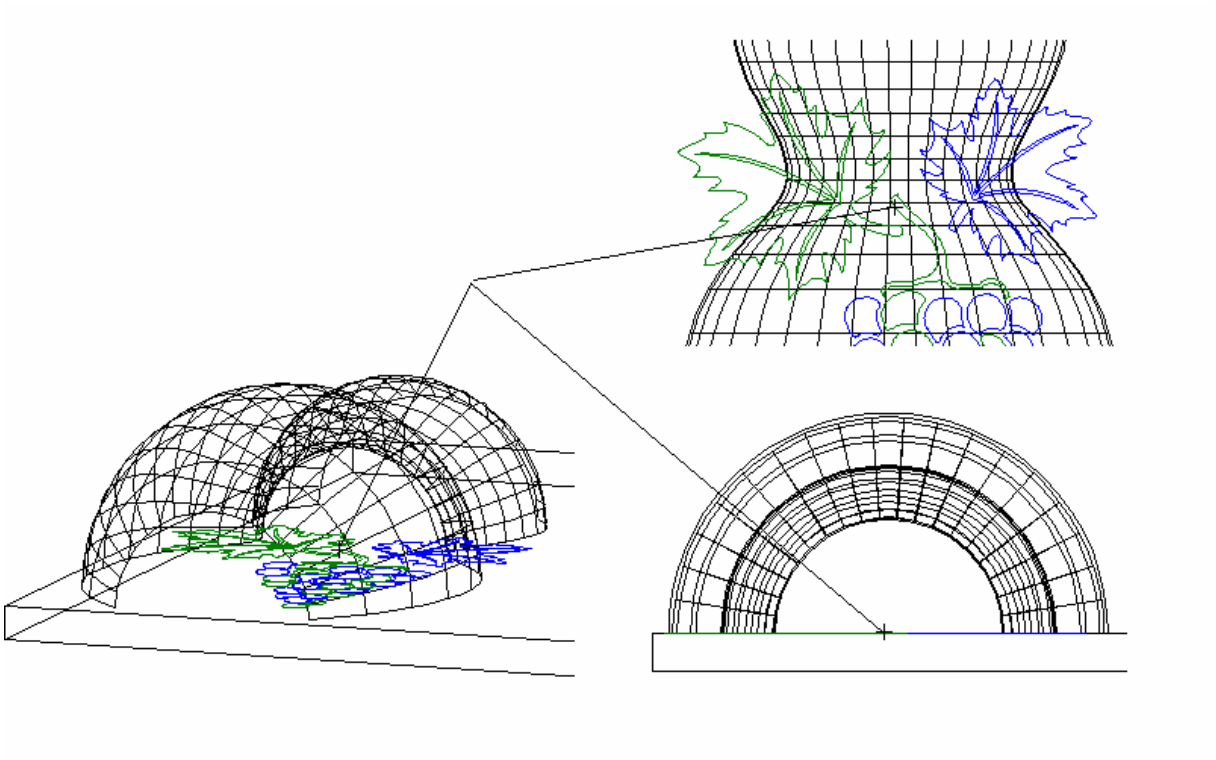
3D ISO view

Basic mapping

REMARK

We consider the mapping is applied to toolpaths, though in some cases it is used directly on curves and patterns (see above Methods and tools for projection and mapping).
If the pattern to map is a text, you should first transform it into curves and combine these curves.

Mapping of toolpaths
Position the toolpath(s) anywhere you would like on the surface (on the X, Y plane) and place a reference point (with a marker) on the center of the design to be mapped.

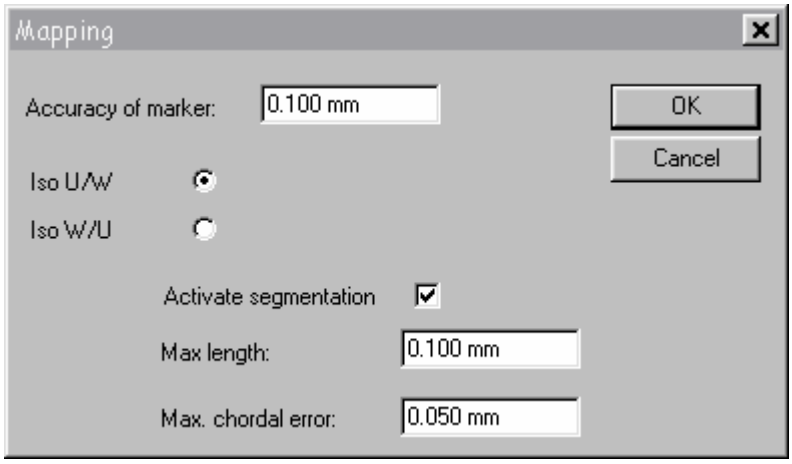


While holding down the **Ctrl** key, first select the path(s), then the reference point and last the surface.

All of them should become red and the icon **Basic mapping** becomes active.

Click now on the icon **Basic mapping** .

The corresponding dialog box is displayed on the screen (the same dialog box can be accessed from the contextual menu of the CAM module).

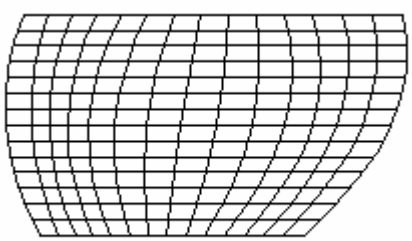


Mapping accuracy
The marker position tells the central spot for mapping the pattern, with an **Accuracy** you specify. The smaller the size of your job and the more important the proportion, the higher your mapping accuracy must be.

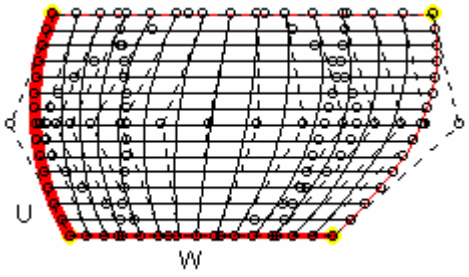
ISO U/W and ISO W/U
U/W are the local axes of the surface on which the drawing will be mapped. With orthonormal guidelines, the U axis corresponds to the X one, and the W axis corresponds to the Y one.

In the first case, axis U (x) is mapped first, then axis W (y). In the second case it is the reverse.
Depending on how complex is the original surface, the warping resulting from the mapping can be different whether the calculation is done first on axis U (then W) or on axis W (then U). You will need to perform some tests, and sometimes invert the curves.

You can see where axes U and W are positioned (often as a result of the way the surface was built) by selecting the design with the modeling tool . U is the thick red axis, W is the thin red axis.



Normal view

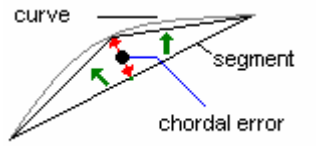


View with modeling tool

Activate segmentation
Option regarding curve segmentation. Activate this option for the mapping. If activated, you can define the accuracy of this segmentation by setting the max. length of the generated segments and the max. chordal error. Generally, the smaller the engraving, the smaller the generated segments and the max. chordal error must be, so that they can be *closer* to the surface.

Max length
Maximum length of any segment (on the X, Y plane). This is the step of calculation before the mapping operation. The smaller your value, the more points you will get and the more time it will take!

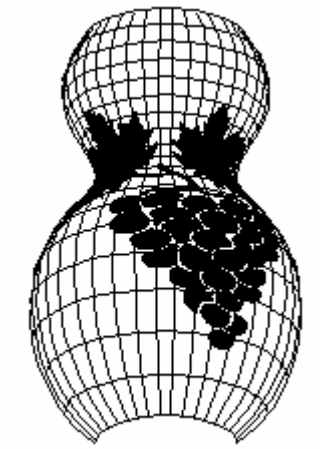
Max chordal error
Clearance between arc and chord (for vectorizing operation in XY plane).



The calculation is done automatically after clicking on **OK**. A progression bar will indicate the status of the job.

To see the result, choose **3D ISO view** from the **View** menu.

3D view before mapping



3D view after mapping

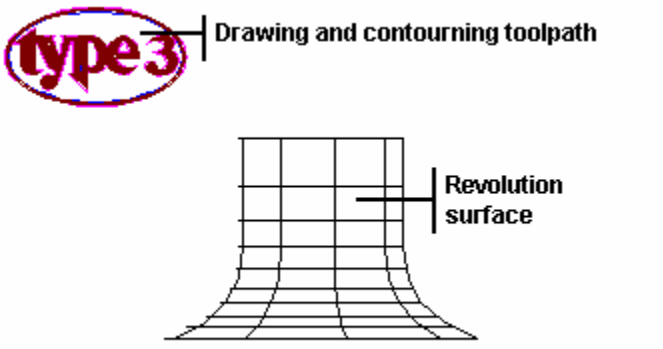
Cylindrical / Conical mapping

REMARK

We consider the mapping is applied to toolpaths, though in some cases it is used directly on curves and patterns (see above Methods and tools for projection and mapping).
If the pattern to map is a text, you should first transform it into curves and combine these curves.

You've got a surface, a drawing and its corresponding toolpath (that is superimposed on the drawing) displayed on the screen.

Example

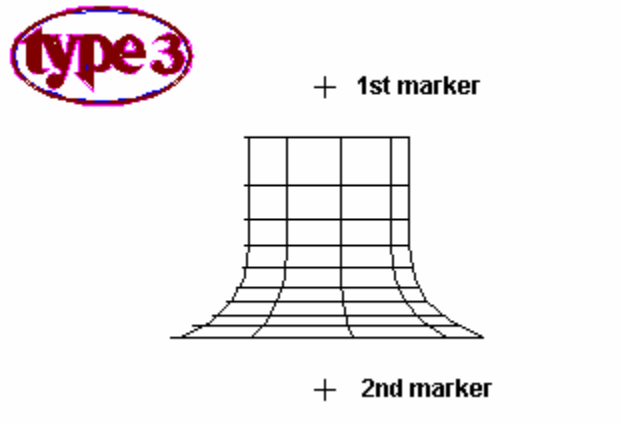


Before mapping, you've got to place two markers defining the cone or cylinder axis. Both correspond to the markers that were used to create the surface.

Remark

The surfaces Cylinder and Cone created with the surface tools are not adapted for mapping.
If you create a revolution surface, select both markers defining the axis (that must be perfectly aligned) at the same time (keep pushed the Ctrl **key therefore) and click on the** Copy **icon** **(you can also hit** Ctrl-C**) before clicking on the icon** Surface of revolution **(actually this last operation will suppress both markers). You only need then to paste the markers on their original place by clicking on the** Paste **icon** **(or** Ctrl-V**).**

Example



Place then another marker for the cylinder radius (two markers for a conical mapping, one for the greatest radius and the other for the smallest radius). Select therefore an **XZ** or **YZ** view from the **View** menu. This marker (or the one of the greatest radius with a cone) will also be used as the *center of mapping* and will define the positioning of the engraving.

Remark

Position the radius marker(s) on the upper part of the cylinder or cone (as in the example below) to make a convex engraving. Position this marker(s) on the lower part of the cylinder or cone to make a concave engraving.

Example

You can also place a reference point (marker) on the engraving (toolpath) to position it on the radius marker (mapping center). Without such a marker, the engraving will be automatically centered according to the radius marker.

Selection order before mapping

Come back to a **XY** view. Select each item by clicking on it while pressing the **Ctrl** key, in the following order.

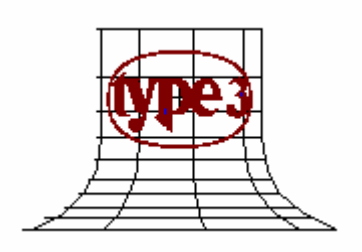
1. the toolpath. Be careful not to select the initial drawing instead.
2. the surface.
3. the two markers defining the axis of the mapping surface (the order of selection of these markers will determine the direction of the engraving.).
4. the radius marker that defines where you want your engraving to be positioned (and perhaps the second radius in case of a cone).
5. the marker used as reference point on the engraving (if this one exists), that will superpose on the radius marker.

Mapping

Once all the elements are selected, click on the icon **Cylindrical mapping** or **Conical mapping** . (This example deals with a cylindrical mapping). The dialog box **Accuracy** is displayed on the screen (see the information in the section **parallel projection**).

Click on **OK** to validate your settings. The engraving is now mapped on the cylindrical part of the surface.

Result



In some cases, among them bottle molds, we advise to use a cylindrical mapping rather than a conical one, because it is easier and causes less distortion of the mapped pattern.

MEASURING TOOLS

Measuring tools

The tools put at your disposal allow you to measure with precision such characteristics as an angle, the distance that separates two contours, the circumference, the perimeter, the surface and the volume of a group of objects. You can also make a tool simulation.

Click on the icon **Measuring tools** to open the measuring toolbar. Click then on the desired icon.

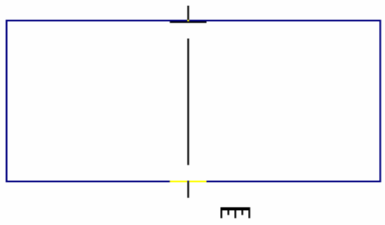
Measuring a distance

Distance

To find the distance separating two points of a contour, after clicking on the icon, select the first point and drag the cursor to the second point.

The distance and the angle of displacement are shown on the inferior status line.

Click on the second point to validate the position. The cursor becomes active on the other side, allowing you to reposition it as many time as you want.



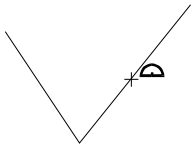
Remark

Used in conjunction with F3, the measurement tool will be more accurate regarding the measured distance.

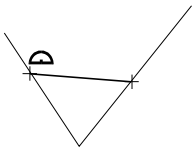
Measuring an angle

Protractor

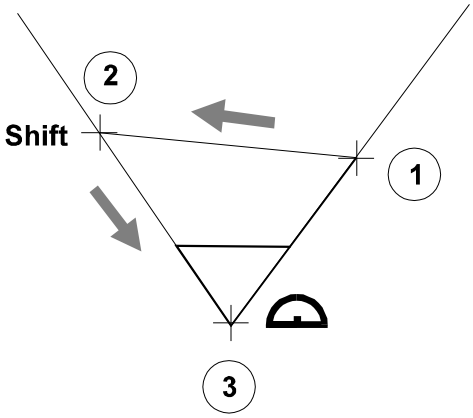
1 - After clicking on the icon, place the cursor along the curve and click on it.



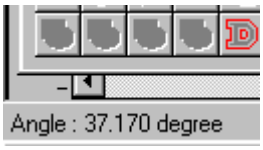
2 - Drag the cursor to the second point and press the **Shift** key (without clicking on the mouse).



3 - While pressing the **Shift** key, move the cursor in the angle you wish to measure.



The value is shown on the bottom status line.



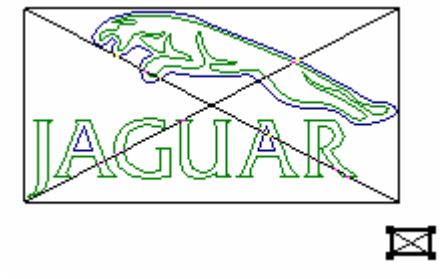
Remark

Used in conjunction with F3**, the measurement tool will be more accurate regarding the measured angle.**

Measuring a size

Size

Click on the **Size** icon. Then place the cursor on the periphery of the object and drag it to the opposite corner, including the object in a crossed square.

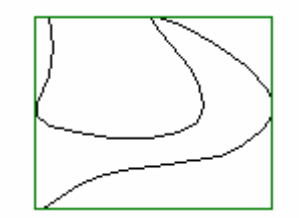


The different values are shown on the bottom status line.

Bounding box

This function allows to measure the real bounds of one curve or several combined curves, even if there are points out of the curve far from the line. A frame, which has the dimensions of the object, is added. The dimensions appear in the status line.

Select the object(s) to be measured and click on the icon **Bounding box** in the **Measurement tools** toolbar.



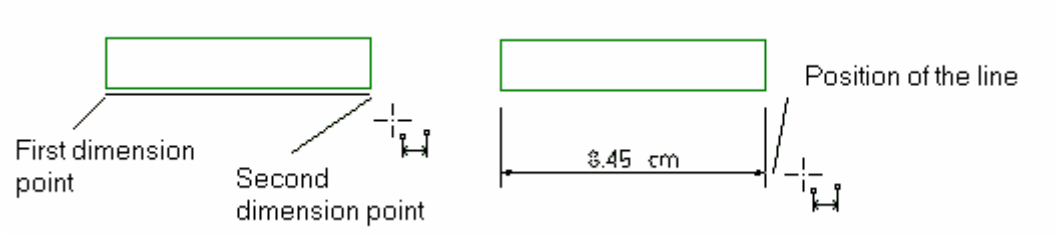
Curve with bounding box

This function enables to apply precise transformations on the object.

Dimension

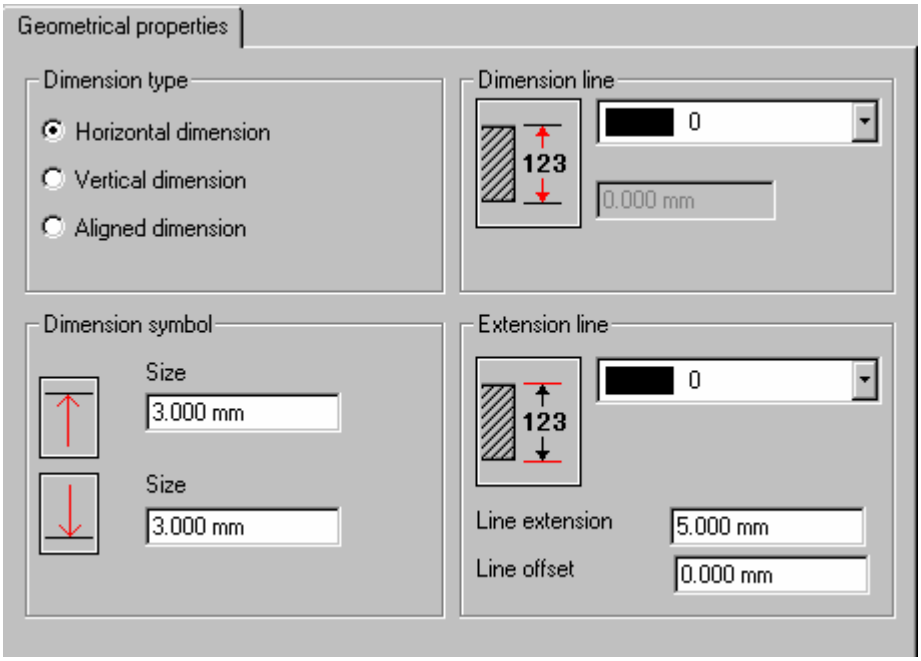
This function allows to add dimension lines to the object and to parameterize their appearance.

Click on the icon **Dimension** . Click on the location of the first point of the dimension line, then, on the second one. Move the mouse in the direction of the dimension lines and click once again when the right position is reached. It is highly recommended to use the control points in the **Snap up** mode (**F3**) to obtain a precise measure.



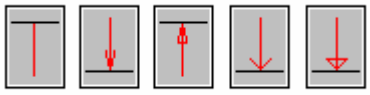
The dimension line is created with default options. A double-click on the existing dimension or a click on **Ctrl** and the icon **Dimension** allow to display the dialog box **Dimension options** with two tabs.

Geometrical properties



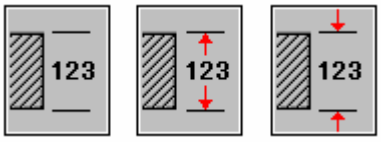
Dimension type
You can choose between 3 types of dimension: horizontal, vertical and aligned. The appearance of the mouse cursor shows, which dimension type you choose.
By pushing the TAB key after having selected the icon **Dimension,** you can change the dimension type.

Dimension symbol
Click on the icon to select amongst five the desired symbol for each end of the dimension line (4 arrows and a line without arrow).



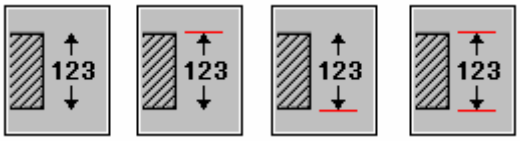
You may enter the size of the arrow in the corresponding field.

Dimension line
Click on the icon in order to choose the position of the dimension line in relation to the extension lines. Three possibilities are at your disposal: without any dimension line, outside or inside the extension lines.



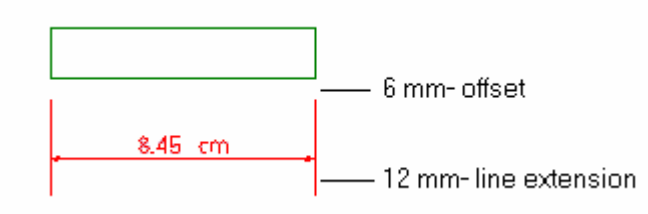
A drop-down menu enables you to choose the color of the dimension line. It is plain that the length of the dimension line must be entered only when it is outside the extension lines.

Extension line
In the same way, you can choose which extension lines are visible or not, and their color.

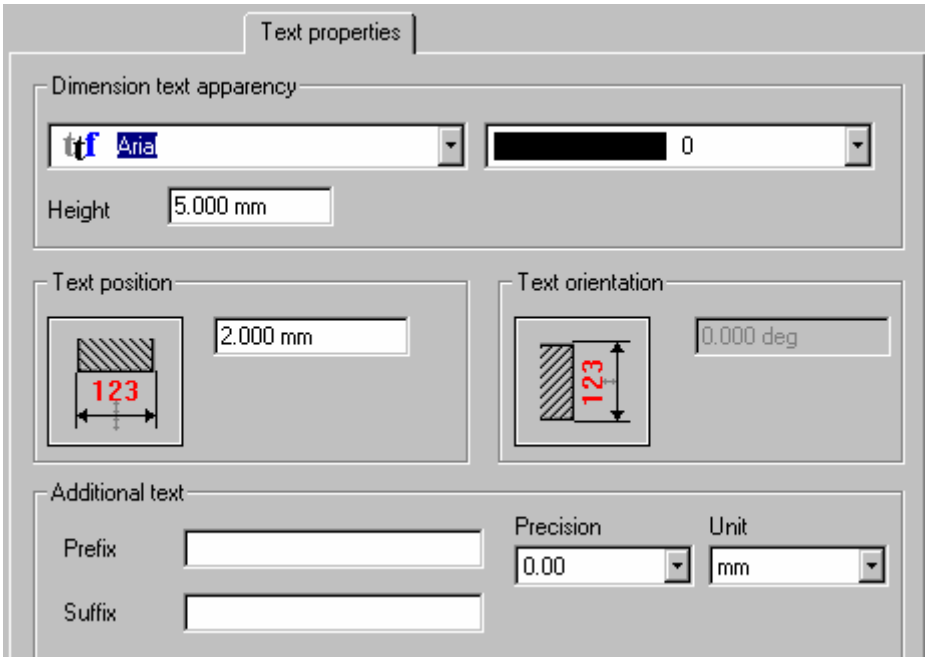


Line extension: Enter the length of the extension line above the dimension line.
Line offset: Indicate how far from the first two dimension points the extension lines have to begin (in general side

of the measured object).



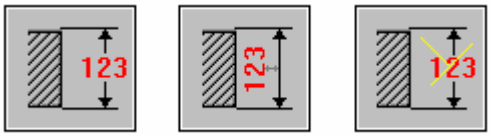
Text properties



Dimension text appearance
Here, you can select the font, the color and the size of the dimension text.

Text position
The dimension text is positioned either on the line, or shifted in relation to this line (the entered value is applied to the baseline of the text).

Text orientation
Choose the orientation of the text (enter an angle value if the text is neither vertical, nor horizontal).



Additional text
Enter the text of a possible prefix or suffix to the dimension text. Please, note that the unit chosen in the drop-down menu is already displayed next to the dimension figure as a suffix.
The number of significant digits (number of zeros after the point) can also be selected in the drop-down menu.

Measuring a perimeter

In the **Measuring** toolbar, click on the icon **Perimeter measurement** .
Place then the cursor on the contour (this one can have points with different heights). Click on the mouse left button. The perimeter value is shown on the bottom status line.

Remark

Used with an open contour, it is like measuring the length of the contour.

Measuring a surface

In the **Measuring** toolbar, click on the icon **Surface measurement** .
Place then the cursor on the contour (it must be a closed contour). Click the mouse left button. The surface value (area) is displayed on the bottom status line.

Measuring a volume

The software includes a tool that allows you to measure the volume of a TypeArt object or even a digitalized file (Picza files with the **.Pix** extension). It's interesting to make use of this tool in the CAD module (white screen) as well as in the TypeArt module (blue screen).

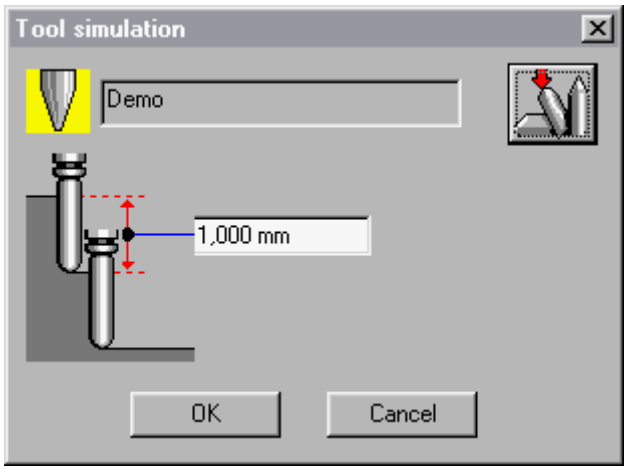
In the **Measuring** toolbar, click on the icon **Volume measurement** .
Place the cursor on the object. Click the mouse left button. The volume value appears in the bottom status line.

Tool simulation

More than a measuring tool, this function permits you to check if the tool, that you will use for the machining fits the created shape.

Before using this function, select one of the 3D views at your disposal in the **View** menu.

In the **Measuring** toolbar, click on the icon **Tool simulation** . The dialog box **Tool simulation** appears on the screen. In this window, click on the icon **Tools DataBase** to select the tool you will use. The dialog box **Select a tool** is displayed. Select your tool and validate by clicking on **OK**.

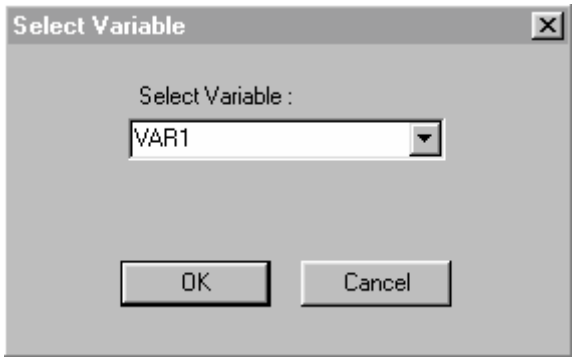


After having selected the tool, click on **OK** in the dialog box **Tool simulation**. The tool is represented by two circles. The little one represents the tip diameter and the big one the half-angle. Drag now the tool cursor in all the desired directions.

Inserting a variable text

You can add a variable text to a composition associated with a data base, directly acquired from the text mode, or imported from a Data Base or a list made in any word processor.

Variable text: Yellow Rapido mode
Type the text and click on the icon where you want to insert the variable text. The dialog box **Select variable** is displayed on the screen.



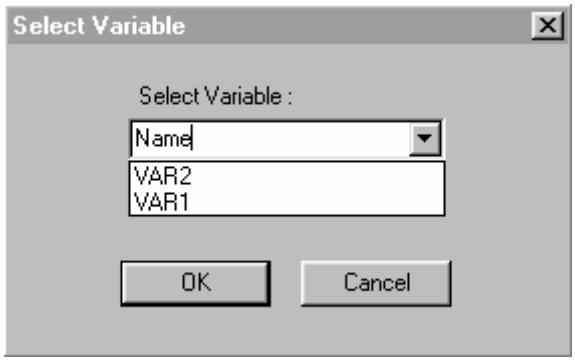
When you insert a variable text, this one is identified by the word **Var** followed by its range number (VAR1 for the first variable, VAR2 for the second one, etc.), and between two bars: **|Var1|**.

Wine: |Var1|
Vine: |Var2|
Vintage: |Var3|

You can add a new variable by:

- selecting the variable name displayed by default (VAR1, VAR2, VAR3, etc.),
- creating a new variable by immediately typing its name in the dialog box **Select variable**,
- selecting an existing variable by using the scroll-down menu in the dialog box.

Example



Remark

Once all variables are created, you can directly modify their name in the text mode. Therefore, you just need to replace the name between the two bars.

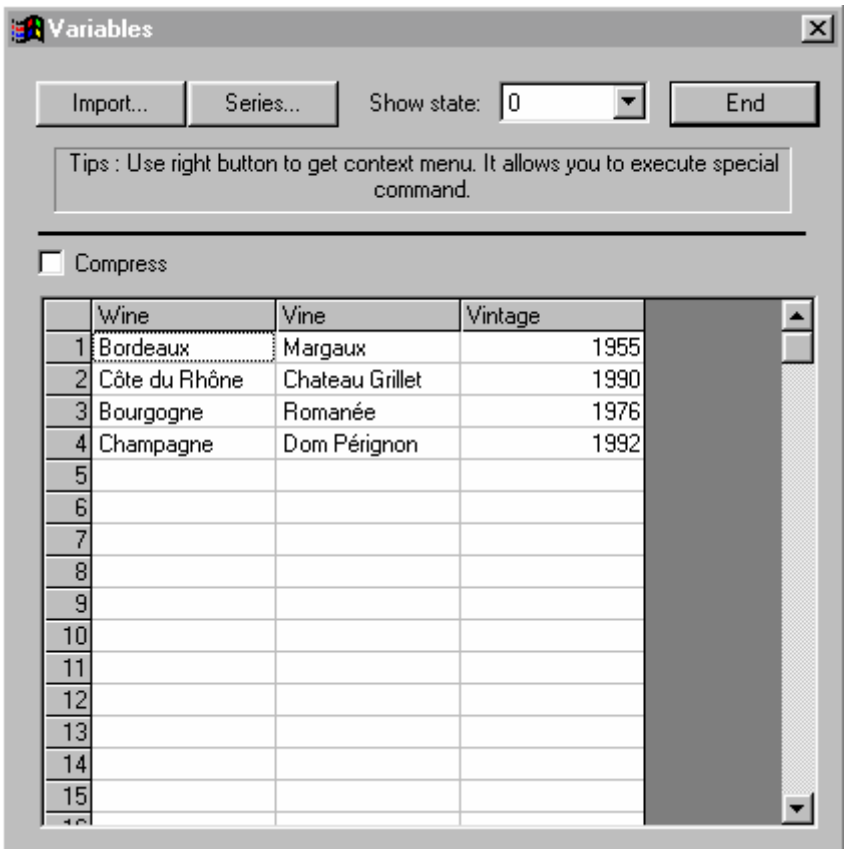
You can modify the attributes (fonts, size, etc.) of a variable text using the same method as for a simple text, and add a box or a logo to your composition.

Once the text is laid out, indicate the data that will replace the variables you just defined.

Saving variable data

Entering variables

Click on the icon **Entering variables** found on the upper status bar.
First method: entering data within the software
Enter in the table of the **Variables** dialog box the possible values for your variables.

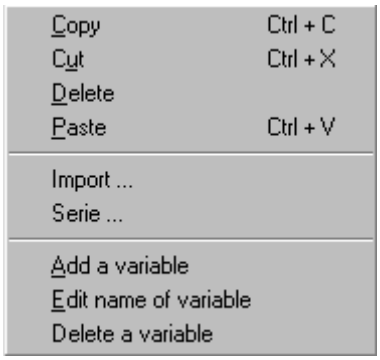


	Wine	Vine	Vintage
1	Bordeaux	Margaux	1955
2	Côte du Rhône	Chateau Grillet	1990
3	Bourgogne	Romanée	1976
4	Champagne	Dom Pérignon	1992
5			
6			
7			
8			
9			
10			
11			
12			
13			
14			
15			

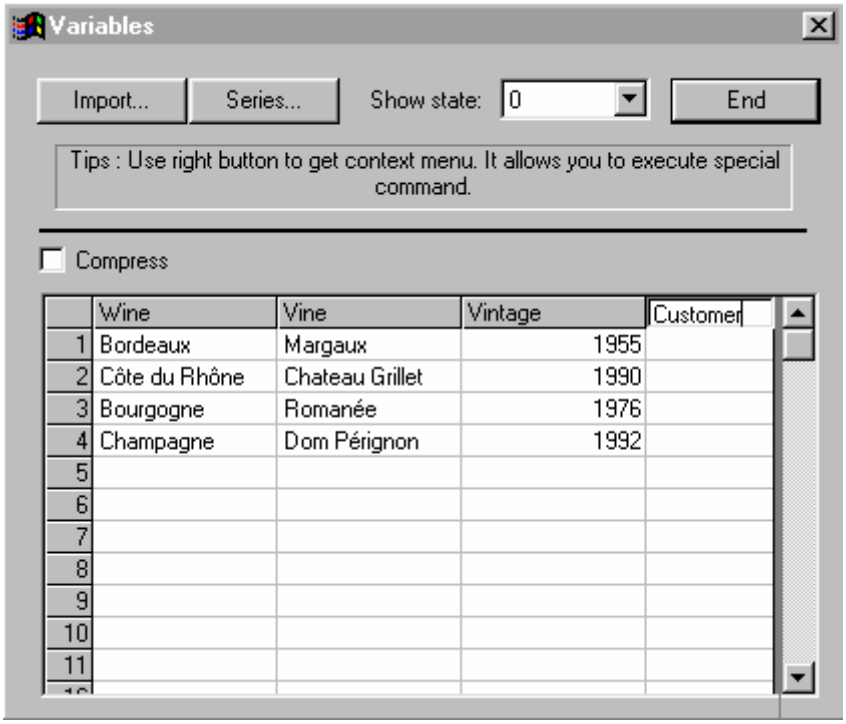
Displaying the data on the screen
Click on the scroll down menu **Show state** to display in your design the actual values of the selected line of variables. Number 1 displays the result corresponding to the first line of variables, number 2 the second, etc.

Adding a variable

You can add one or several variables during the definition of the variables. Click anywhere on the dialog box **Variables** with the right button of your mouse. The contextual menu is displayed.

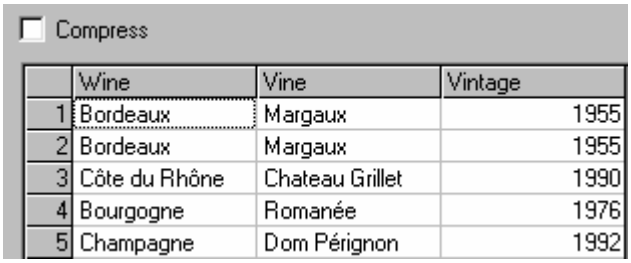


Click on **Add a variable**. Define then the name of the variable and its values in the appropriate fields.



Compression mode
This mode is useful in case of identical variables that follow each other.

Example

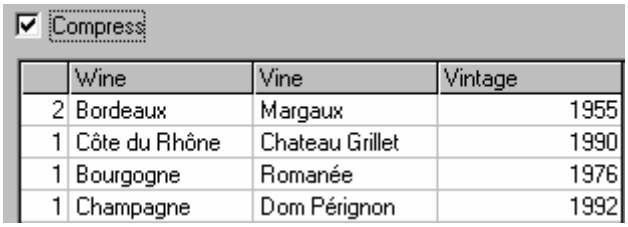

Compress
Wine Vine Vintage
1 Bordeaux Margaux 1955
2 Bordeaux Margaux 1955
3 Côte du Rhône Chateau Grillet 1990
4 Bourgogne Romanée 1976
5 Champagne Dom Pérignon 1992

In the example shown above, the lines 1 and 2 contain exactly the same variables (Wine: Bordeaux, Vine: Margaux, Vintage: 1955).

NOTE
Even the smallest difference (Character or space) between two lines keeps from using this function. Moreover, similar values must be consecutive to be considered in a same group.

Select **Compress**. The **Variables** dialog box automatically groups the identical variables.

Example



	Wine	Vine	Vintage
2	Bordeaux	Margaux	1955
1	Côte du Rhône	Chateau Grillet	1990
1	Bourgogne	Romanée	1976
1	Champagne	Dom Pérignon	1992

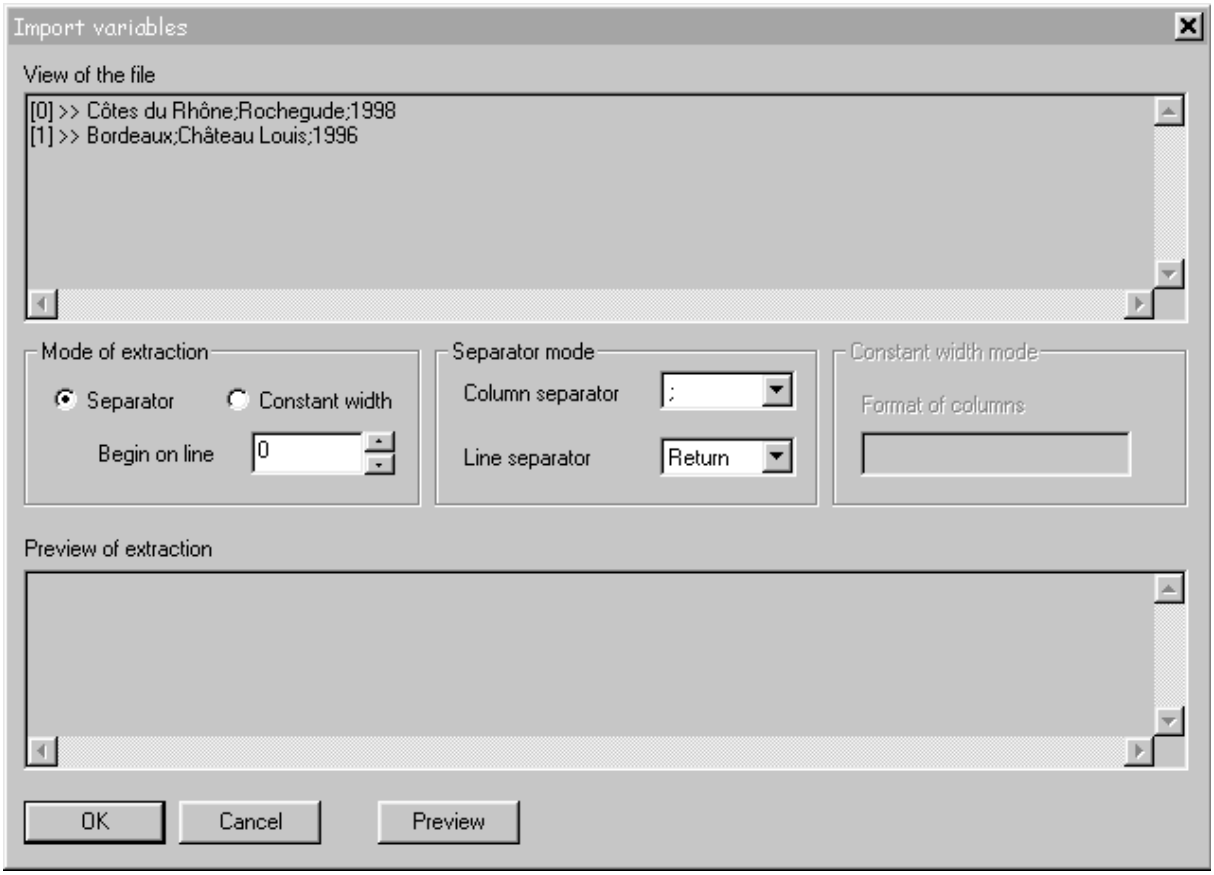
The line numbers are not displayed anymore. The number before each variable indicates its frequency.

Importing variable data

You can get data from any word processor, data base or CAD program (ASCII or "text only" format, in general with a .TXT extension).
A single character is used to separate the values of different variables (in general a semicolon or tabulation). Records are usually separated by a carriage return.

After having created the variables, choose the **Import** command from the **Variables** dialog box.

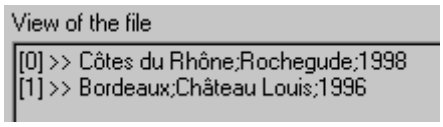
Search and select the file to import (*.TXT), then click on **Open**. The data appear in the **Import variables** box.



The extraction mode
The **Import variables** dialog box allows to define the beginning of the extraction and the mode used for extracting data. The dialog box features two extraction modes: **Separator** and **Constant width**.

Begin on line
Select from which line in the file the extraction must be done (0 to start from the beginning). The line numbers are displayed in the **View of the file** section, between brackets: [0] and [1] in the example below.

Example:



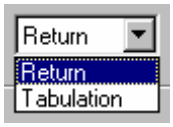
The choice is made in the **Begin on line** menu.

In the example shown above, the extraction begins at line **[0]** (Côtes du Rhône;Rochegude;1998)

Separator
This mode is the most common and allows fields in which the size of the values changes a lot between different variables. Of course, you must have defined characters or instructions that will separate the different fields.
Click on **Separator** and select the column separator using the first scroll-down menu in the **Separator mode** section.

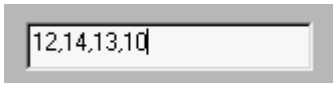
For the example used in this section, "**;**" must be selected.

Select then the line separator using the second scroll-down menu. For the example, select **Return**.



Constant width
Selecting this mode, you define the width of the different columns containing the variables.
Click on **Constant width** and fill in the **Format of columns** field. The numerical value you enter in that field corresponds to the number of characters contained in the column. In case fields have different widths, enter the values for each of them respecting their order of display. In that case, separate all the values with comas.

Example



In the example, the first column can contain 12 characters, the second column 14, the third column 13 and the fourth column 10.

Preview of extraction
You can check if the setup of the extraction was properly carried out before validating it and quitting the **Import variables** dialog box. Therefore, click on **Preview**. The result is displayed in the **Preview of extraction** field.

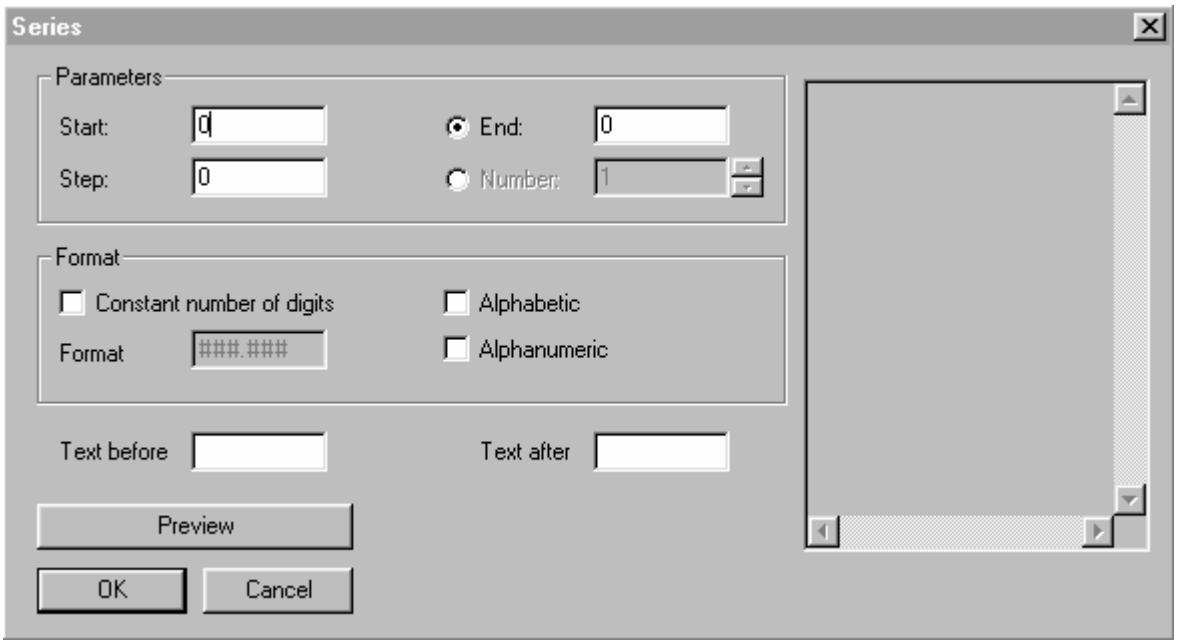
Once the extraction setup is made, click on **OK** to validate.

Creating a serialization

You can automatically create an automatic set of numbers in a variable text: the serialization.

Variable text (Yellow Rapido)
Enter the variable at the desired place in the composition.
The variable **Var** appears between two bars.

Entering a serialization
Click on the icon. The **Variables** dialog box appears.
Click on the **Series** button after having selected the column you want to apply this function to.



Enter in the **Parameters** field of the **Series** dialog box:

- the starting number
- the step between each series
- the end value, or similarly the total number of values (size of the series)

Example: Numbering of hotel rooms, even numbers only from 18 to 26.

- Starting number: the first room will have the n° 18.
- Number of series: 18, 20, 22, 24, 26; in other words 5 series, or ending value of 26.
- The step: for a numbering in even numbers, 2.

Format
The software considers by default that you create a numerical series. The **Format** section allows to chose other data types: letters (Alphabetic) or numbers + letters (Alphanumeric = an "alphabet" of 36 signs).

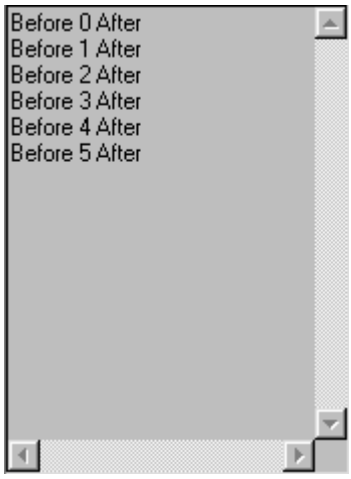
You can also chose some formatting for display, to have a constant width whatever are the actual values in the series. Check the **Constant number of digits** box and edit the suggested format (each # character will be replaced by a sign).

Added text
The dialog box **Series** allows you to add text before and after the variables. Enter the text in the **Text before** and **Text after** fields of the dialog box.

Remark

To include a space before and/or after the variable value, you must type this space when you enter the text in the fields.

Click on **Preview** to display your series before quitting the **Series** dialog box.



Remark

You can enter as many different series as you wish in the same data base.

Example: *the row 1 corresponds to room 18.*

Remark

For the variable text, you can also use the copy/paste **function of Windows.**

MULTICOPY AND MULTIPLATE MACHINING

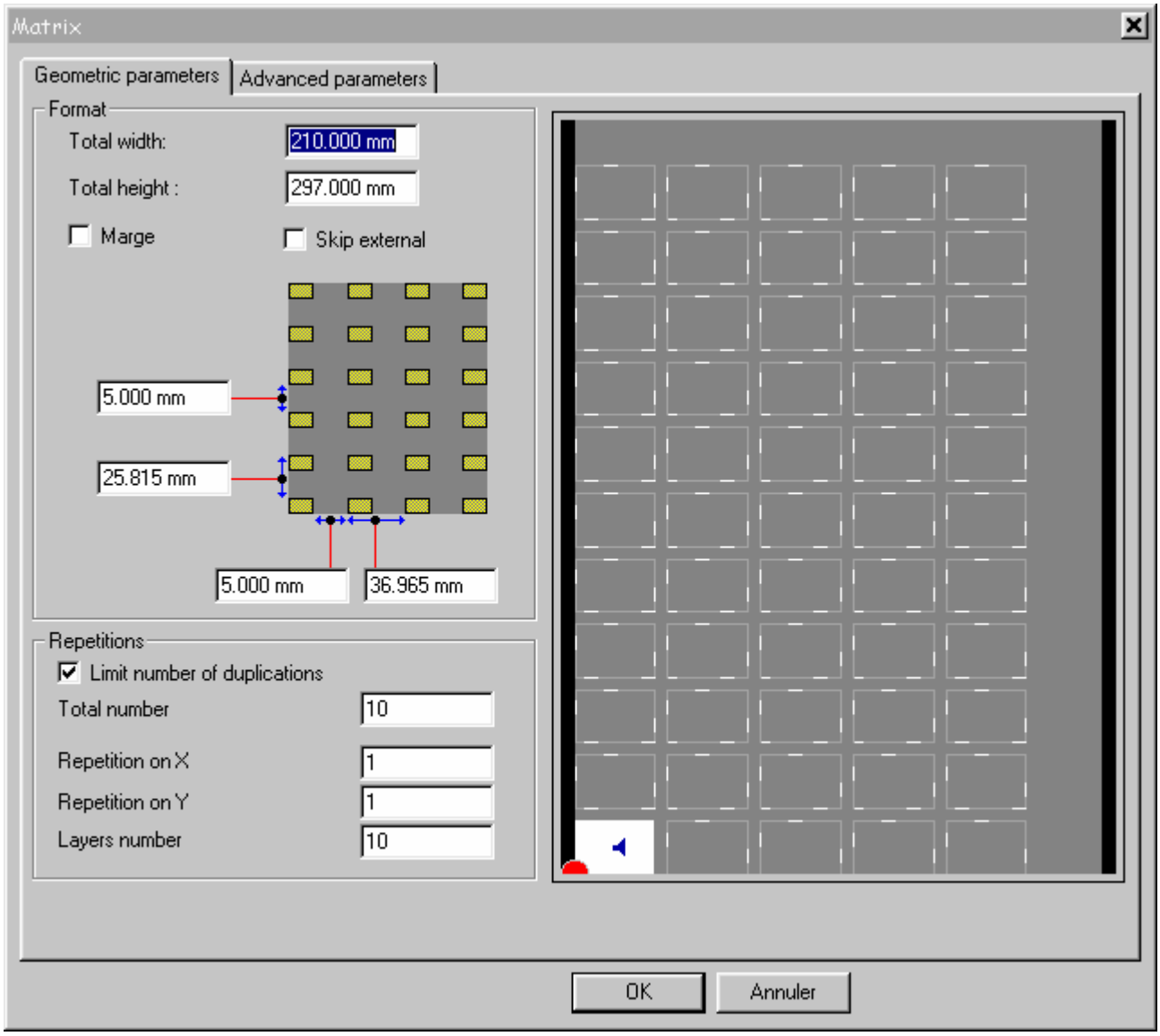
Introduction

The multicopy function on the toolbar generates automatically copies of graphics, texts and variables.

Multicopy
Two modes of multicopy are at your disposal:

- several copies on a plate,
- one copy on several plates.

Select the objects you want to reproduce several times and click on the **Multicopy** icon.



Multicopy: geometrical parameters

Dimensions of the plate

In the **Total width** and **Total height** fields, found in the **Multicopy** window, enter the dimensions of the plate to be machined or the dimensions of the machine engraving surface.

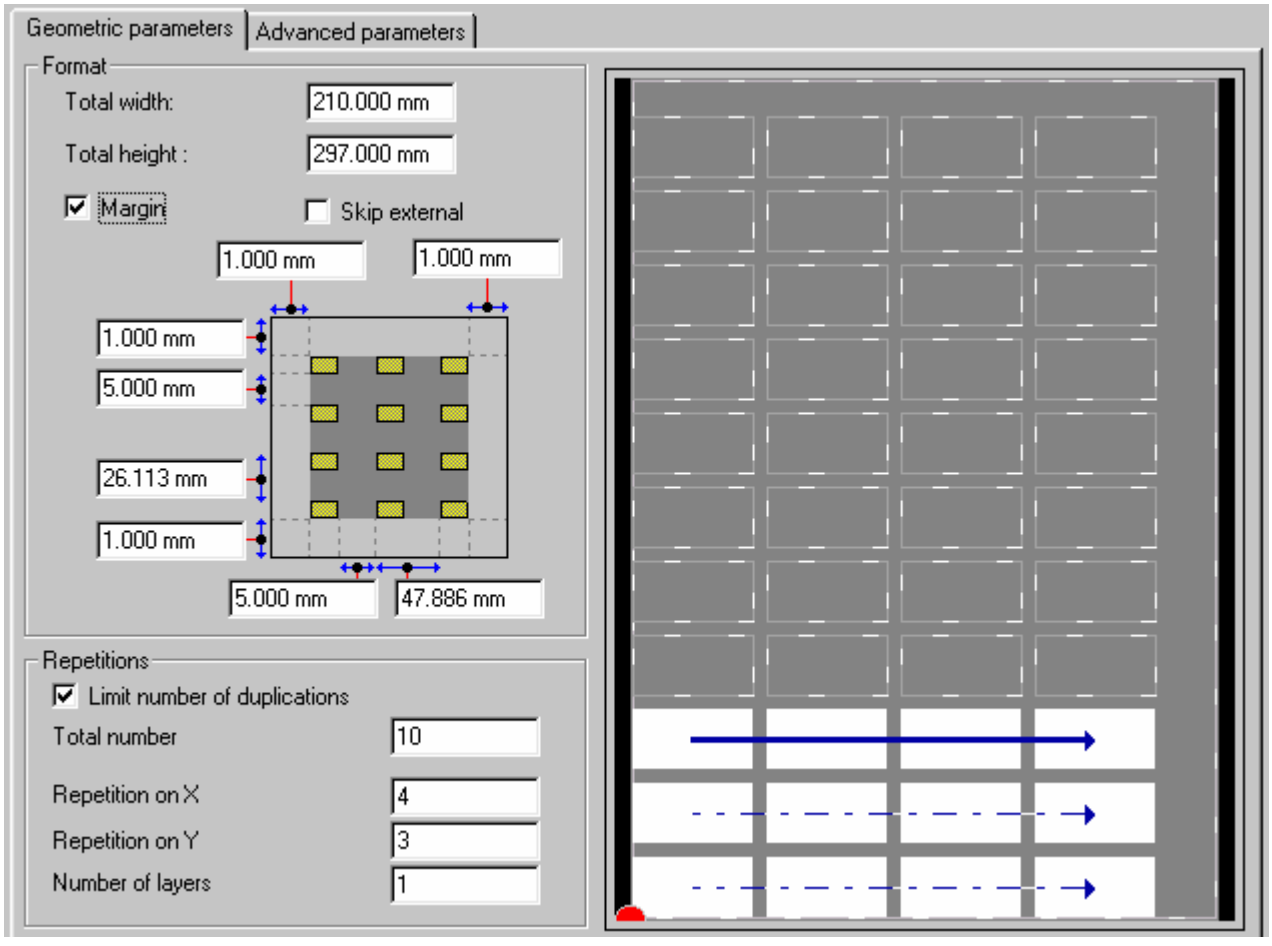
Remark

In order to see the preview window for the copies, the dimensions of the machine engraving surface have to be greater than those of the active surface.

According to the given information, the display of the field to the right of the window determines the possible number of copies.

The white square situated at the top of the preview window corresponds to the area of the original object on the plate.

The gray squares indicate the possible number of copies, according to the dimensions of the plate to be duplicated and to the dimensions of the machine engraving surface (interval between the border and the spacing between the copies included).



In the **Repetitions** section, enter the **Total Number** of duplications for your object. According to the size of the element, the size of the plate and the spaces specified, the software calculates the number of repetitions in X and Y. If the plate is filled but the total number of duplications is not reached, the **Number of layers** is incremented.

Alternatively, type the number of copies in the fields **Repetition on X** and **Repetition on Y** (rows and columns). With the mouse, you can also click on the gray squares to validate the position and the number of copies required.

In case where machining on several different plates, it is recommended to adapt the size of the active surface to the size of the plate to be machined.

In this case, machining is done plate/plate, a stop code is sent to the machine to allow you to change the plate during the machining process.

For machining variables, the **Nb of layers** field is limited to the number of variables. For example: 20 variables data = 20 layers (maximum).

Remark

For the creation and import of variables data, see the previous section about Variables.

The other parameters should not be modified in the case of one copy on several layers, since they are reserved for machining several copies on one layer.

Format

In the **Format** section, enter the desired distance between each object (e.g.: cutting line), which will calculate automatically the distance between each starting point of objet. You can also do the reverse. You can use or not a margin (value corresponding to the distance between the copies and the border of the machine table).

When you leave the dialog box by clicking on OK, on the screen, the copies are superimposed, one on top of the other.

Remark

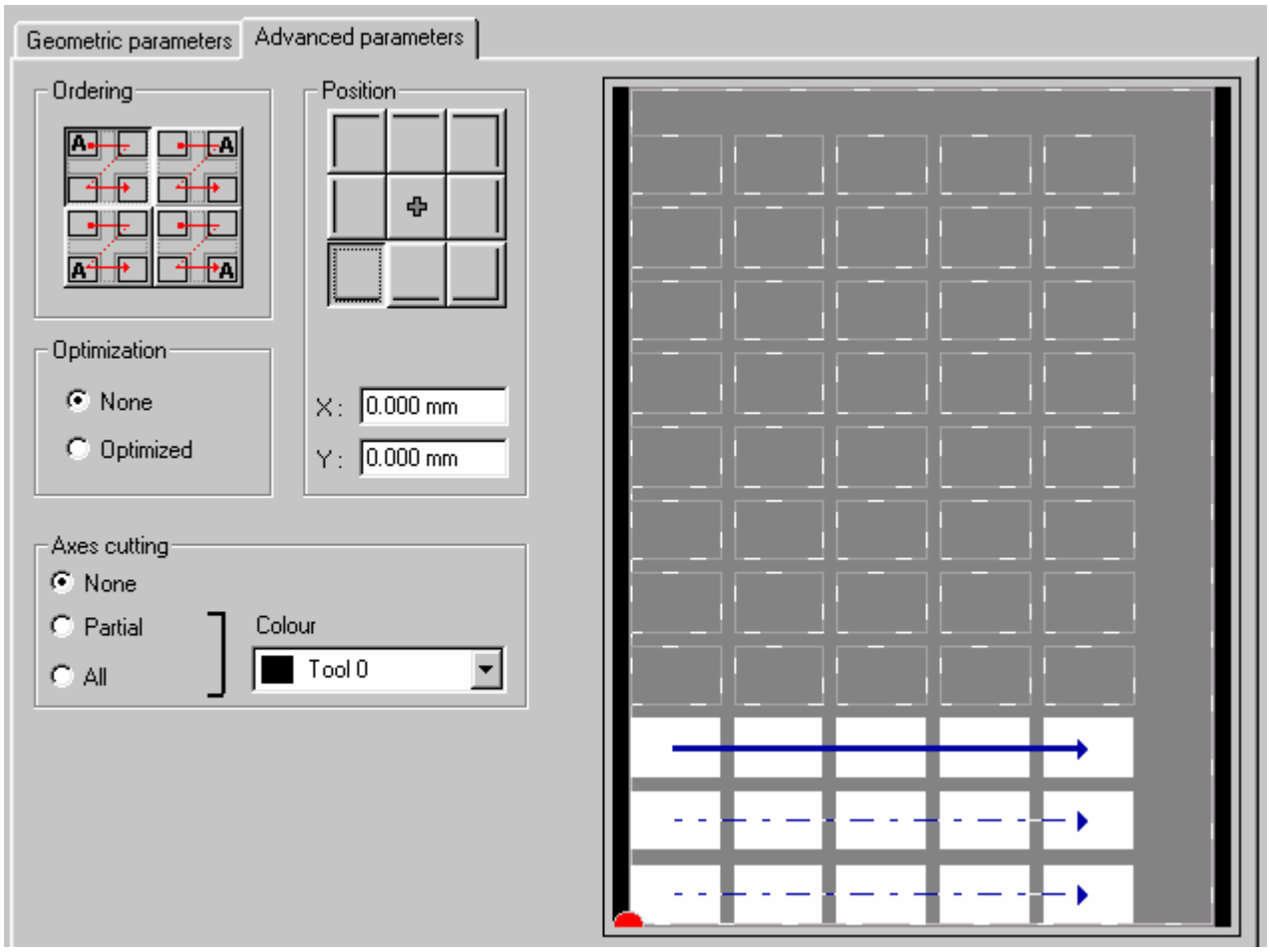
The original drawing is included in the total number of copies.

Therefore, when validating a multicopy, the software automatically generates the necessary number of layers as follows:

- **Layer 1 for the first layer,**
- **Noname + the number from 0 to the number of layers selected - 2 for the n following layers.**

Multicopy: advanced parameters

The **Advanced parameters** tab is used to specify the options related to the machining of the plates.



Ordering

The **Ordering** section allows you to specify the machining order of the items on the plate. It is used when the items include numbers that are incremented and that should be obtained in a specific order.

But if the radio button **Optimization** is **active**, the software will decide of the machining order, giving priority to the least move.

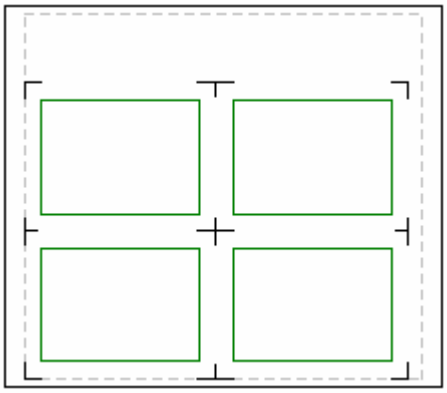
Position of the first element

The **Position** section allows you, by moving the red spot with the mouse or by entering precise coordinates, to define the position on the plate of the first item machined.

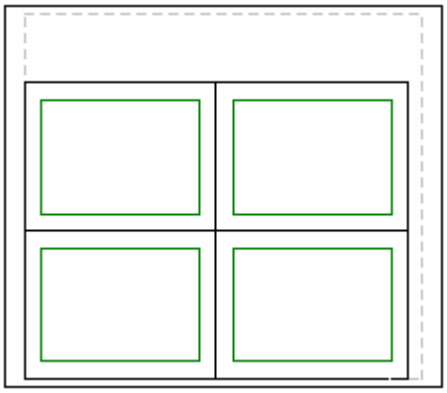
Axes cutting

To help you with the cutting of the plate, you can add partial or total guidelines between each copy.

To validate you choice, select the desired option: **None**, **Partial**, **All**.



Partial cutting



Total cutting

Remark

The depth of cutting of the cutting lines can be modified in the Machining **window.**

You can chose a specific tool for the cutting between each copy: associate a color in the Axes cutting section. Use then the Color machining icon , after having selected the relevant contour: you will find again the chosen color, and with a simple click (not a double-click), associate to it a tool from the tool database.

SCANNING

Running the scanner

Scanner
The scanner allows you to copy a printed drawing.
If you have a scanner installed that is compatible with **TWAIN** you can scan and vectorize directly with the CAD module.
When you click on the icon, the image appears in the active area.

Remark
Click only once on the icon!
Give time to your scanner to begin working.

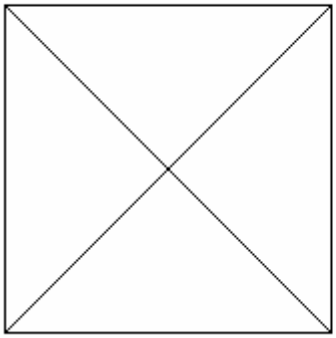
Click on the tool. The command menu of the vectorizing software provided with your scanner will be displayed on the screen. The CAD module will display the raster Bitmap image. Raster images are composed of fixed size pixels (small black and white points that make up the picture).

The quality of the reproduction depends on the quality of:

- the drawing,
- the scanner,
- the chosen settings (brightness, contrast, zoom),
- the accuracy required (DPI resolution).

Visualization of the Bitmap

To display the image, select **Options** in the **Edit** menu. In the dialog box **Options**, choose **Display** and select **Show bitmap**.
If you do not want to display the points generated by your scanner, a frame will appear instead of the drawing. This visualization is advised for large drawings which would delay the display. Moves and displays will go much faster thereafter.



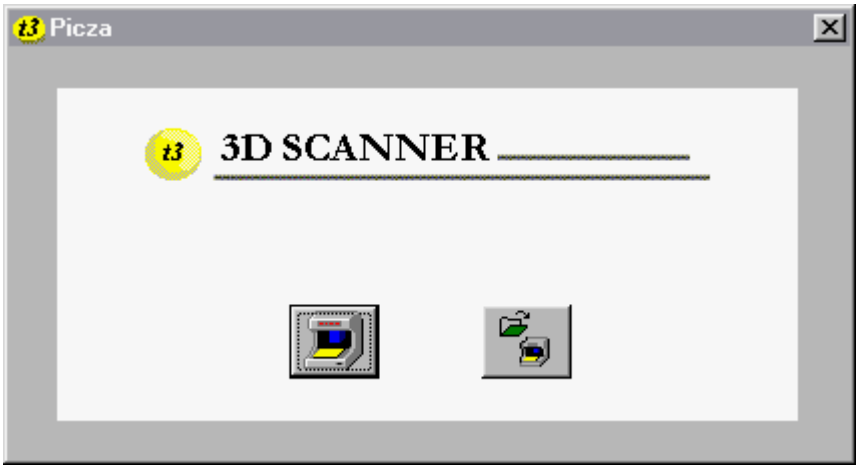
Fast display
(Show bitmap option deactivated)

Real display
(Show bitmap option activated)

Scanning in 3D

This option allows you to scan in 3D (digitalize) a surface or an object using the **Picza** module or to import a file generated by the same module (both are **Picza** files named ***.Pix** files). Such surfaces can be machined as a **TypeArt** file.

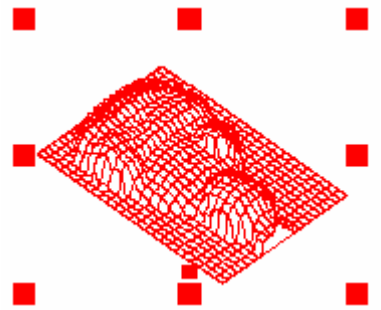
Scan 3D
Click on the icon **Scan 3D**. The dialog box **Picza** is displayed on the screen.



Acquire with Dr Picza
Click on this icon to scan in 3D an object or a surface with the **Picza** module. Please refer to the **Picza** manual edited by the Roland Corporation for any information about the use of this 3D scanner.

Open Picza files
Click on this icon to load already existing **Picza** files (***.pix** files) in **TypeEdit**. The digitalized **Picza** file appears like a **TypeArt** drawing on the screen. As you send it to the CAM module, you can only compute **TypeArt** toolpaths.

*Example of a **Picza** file the way it appears in the software*



VECTORIZING

Running the vectorization

The vectorization module converts raster images into vector images, that can then be machined.

Vectorizing
Select or import the image to be vectorized in Bitmap format (*.TIF or BMP) and click on the vectorizing icon. The vectorization is done automatically.

Click only once on the icon!
Give your scanner time to begin working.

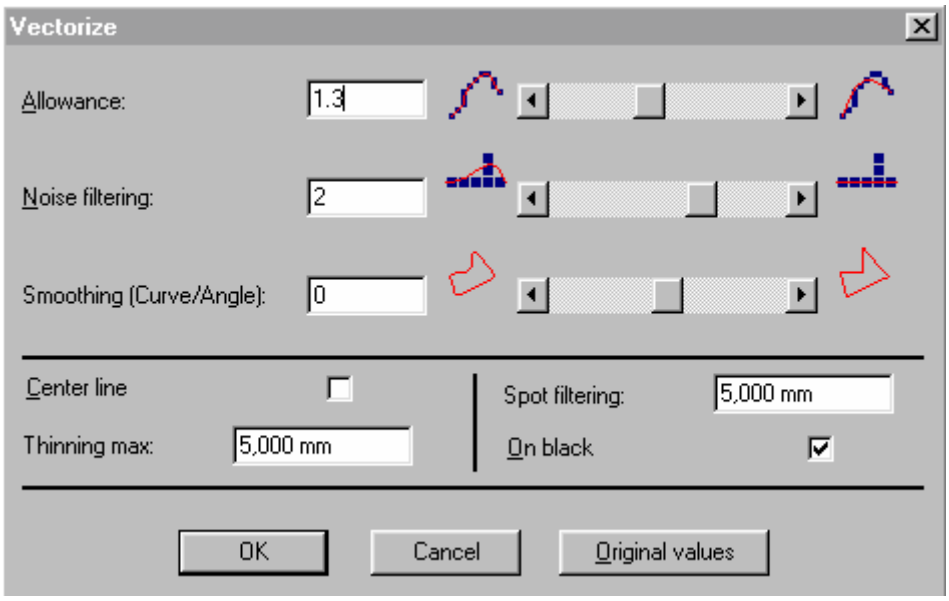
Remark

If no image is selected, the vectorization mode will automatically begin, and will scan and vectorize the image in one continuous operation.

This function is useful to directly retrieve the contours of the scanned image without the bitmap image in the background.

Changing the parameters of vectorization

To modify the parameters of vectorization go into **File → Vectorization parameters…,** or click on the icon while simultaneously pressing **Ctrl**.



Allowance

from 0,1 to 3

This parameter determines how faithful the resulting image will be to the original image.

Higher values lead to smoother vector image while lower values make the vector image fit more accurately to the actual pixels in the raster image.
The better the quality of the scanned drawing, the better the result, of course…

Be careful with this setup because the demanded precision has a direct influence on the number of points generated!
The points can be overabundant and slow down the display on the screen, the manipulations, and the calculations.

Noise filtering

from 0 to 3

This parameter determines how to treat the small noise pixels that appear in an image. A small value results in a more detailed vector image. A larger value leads to a smoother image, less faithful to the original raster image.
This option is very useful for avoiding future corrections on final curves.

Smoothing

from -1 to + 1

This parameter determines the choice of vectorization using either a conversion in splines or segments, or favoring the former more than the latter or vice versa.
-1: all in splines
+1: all in segments

At 0, all the angles under 90° become sharp.

Centerline
Selection of the type of vectorization (contour or skeleton).

Bitmap *Contours* *Skeleton / Centerline*

Maximum thinning
Maximum value allowed for thinning contours for the vectorization in centerline mode.
The calculation is made in pixels in function of the image resolution and of the width of the original tracing of the drawing.
This parameter should be approximately equal to half of the number of pixels relative to the widest tracing of the original drawing to be vectorized.

Spot filtering
Periphery in number of pixels of the spots to be deleted.
This option is very useful when the drawing is of bad quality. It limits the number of contours created during the vectorization by ignoring the elements considered as not being part of the drawing.

On black
☒ Vectorization is made on the black part of the bitmap.
☐ Vectorization is made on the white part of the bitmap.

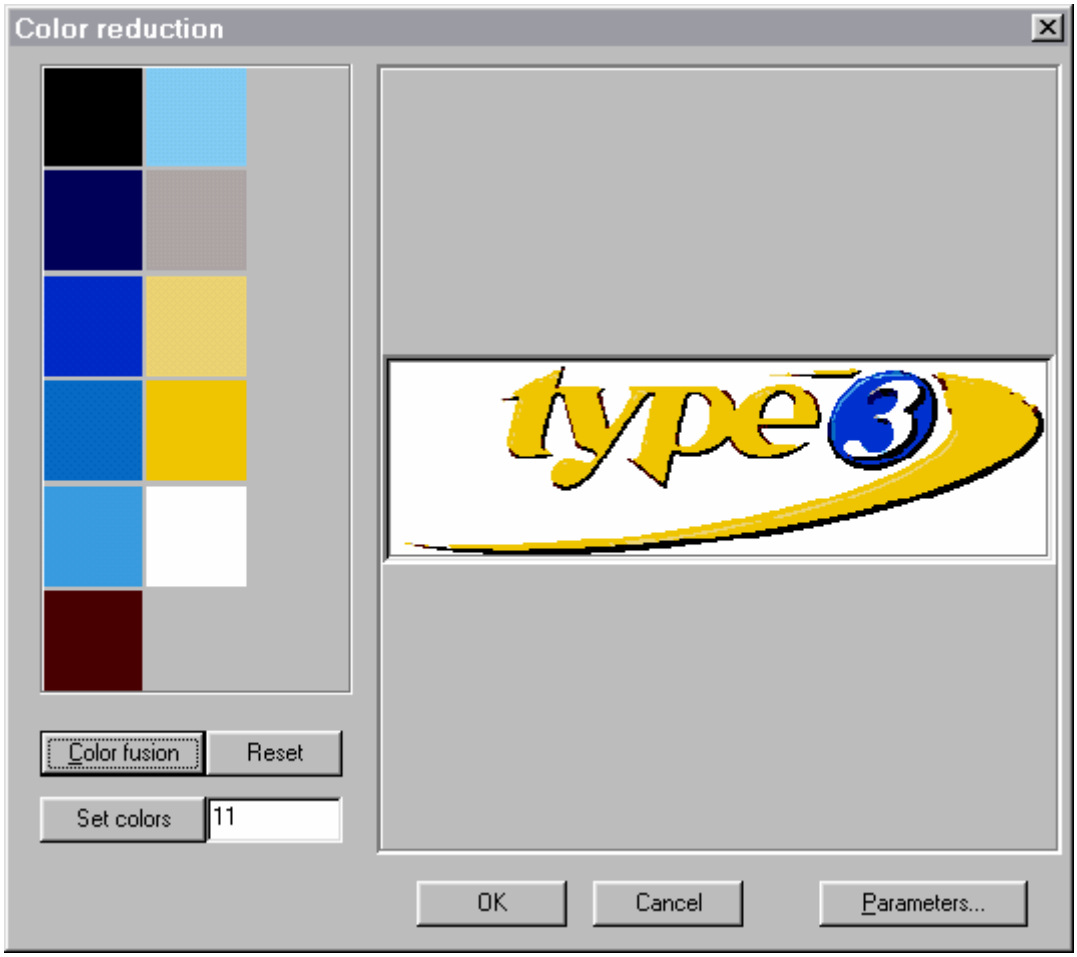
Default values
Allows you to return to the standard set values.

Vectorizing color bitmaps

The vectorization module can also convert raster color images into vector color images, which can then be machined.

Vectorize bitmap
Select or import the color image to be vectorized in Bitmap format (***.tif** or **bmp**). If you want to modify the parameters of the vectorization, strike the **Ctrl** key while you click on the icon (or choose Vectorization parameters in the **File** menu): see therefore the **Changing the parameters of vectorization** section.

Click then on the **Vectorize bitmap** icon. The dialog box **Color Reduction** is displayed after a few seconds of calculation.



In the example shown above, the module automatically selected five different colors that will be used for running the vectorization.

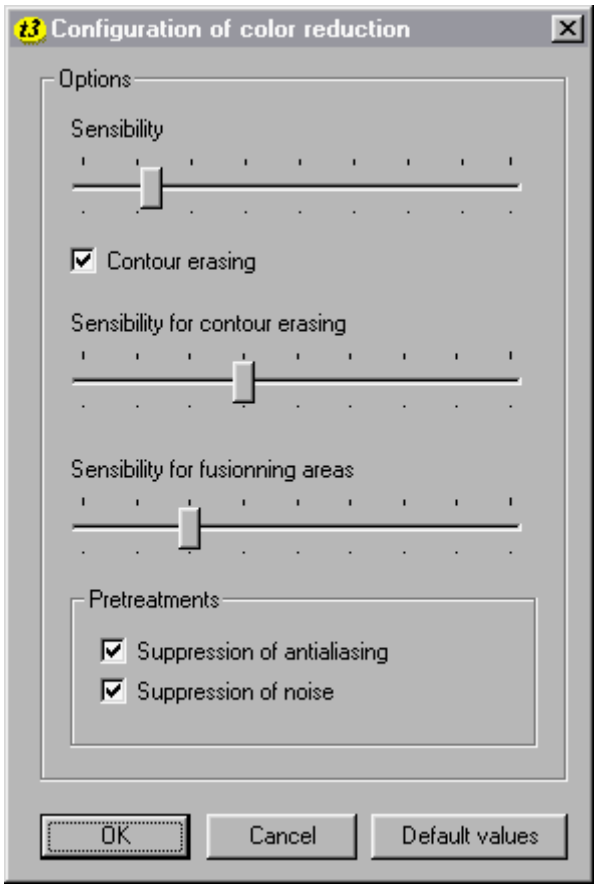
Nevertheless, the module allows you to modify the automatic settings of the color reduction (color fusion and number of colors) which is the first step of the vectorization process.

Color fusion
Each color which has been selected by the vectorization module is represented by a square in the left part of the dialog box **Color reduction**. You can select several colors you want to get fusioned by clicking on each corresponding square while keeping pressed the **Ctrl** key. Once you selected the desired colors, click on the **Color fusion** icon. The resulting number of colors automatically appears in the field located at the right side of the **Set colors** icon.
You can get back the previous settings by clicking the **Reset** icon.

Set colors
You can set this value to be exactly the number of colors you want to apply to the vectorization. Enter therefore the desired value and click on the **Set colors** icon. This operation may last a few seconds depending on the amount of colors you want to use.
You can get back the previous settings by clicking the **Reset** icon.

Parameters

You can parametrize the configuration of the color reduction by clicking on the **Parameters** icon. The dialog box **Configuration of color reduction** is displayed on the screen.



Pretreatments

Suppression of antialiasing
Activate this function to keep the sharpness of the angles of a contour.

Suppression of noise
Activate this option to suppress the small noise pixels that appear in an image.
This option is very useful for avoiding future corrections on final curves.

Remark

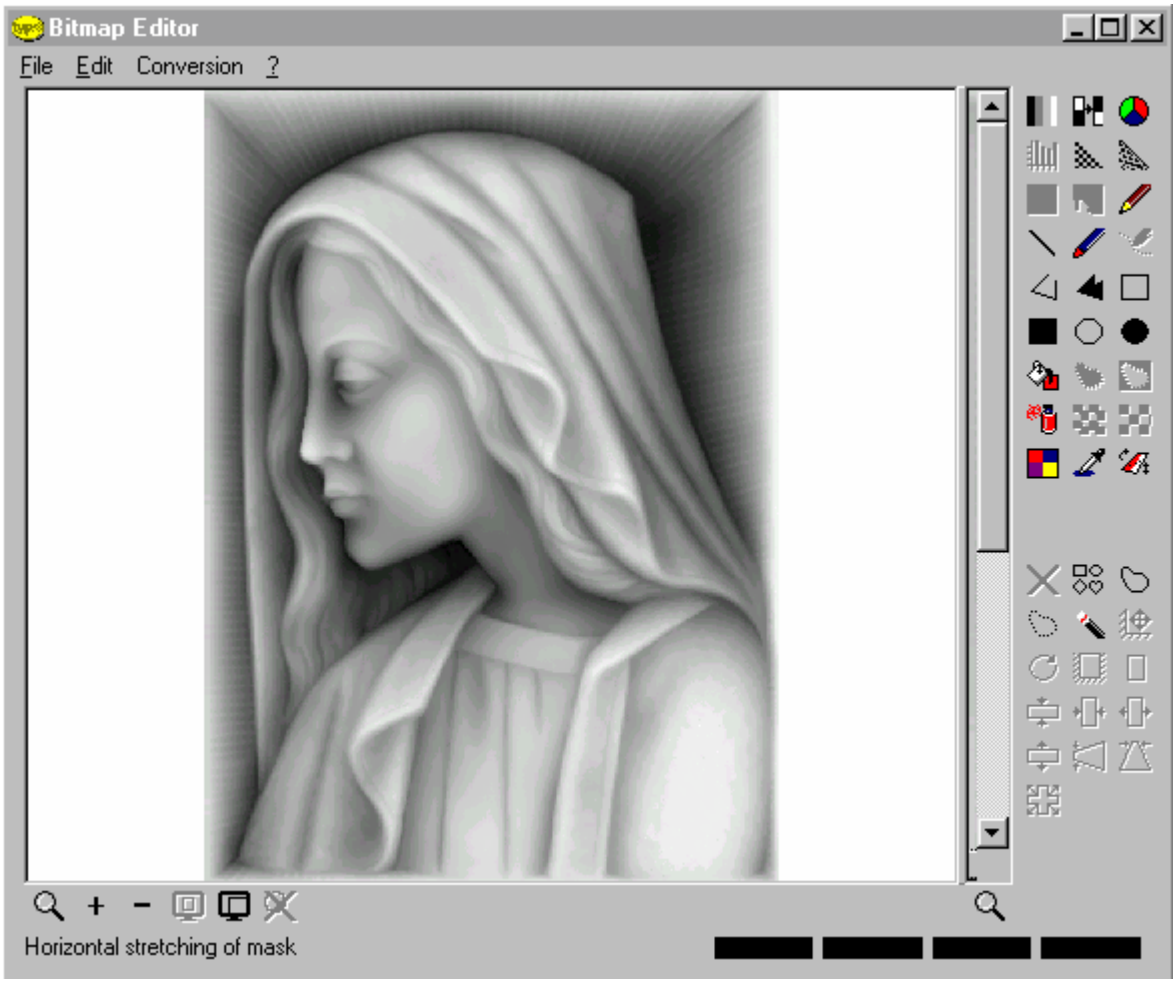
If you click on the icon Default values **you will get back the values that were automatically determined by the vectorization module.**

BITMAP EDITOR

Running the bitmap editor

Bitmap editor

Click on the **Bitmap editor** icon to launch this function once you have loaded a bitmap in the CAD module.
The dialog box **Bitmap editor** is displayed on your screen.



Working with images

Selecting the active color

Bar of the four last selected colors
Click on one of the four last selected colors that are represented by four rectangles in the lower part of the dialog box **Bitmap editor**.

Eyedropper
Click on the icon **Eyedropper**. Place then the eyedropper on the color you want to be selected and click on it. The selected color becomes the active one and is displayed in the bar of the four last selected colors.

Color selection (for 256 color images)
Click on the icon **Color selection**. The dialog box **Choose a color in palette** appears on the screen. Click on the desired color in the dialog box with the left button of the mouse. The selected color appears in the bar of the four last selected colors.
In case you want to edit the selected color, just click on it with the right button of the mouse in the dialog box **Choose a color in palette** (it opens the dialog box **Color edition**).

Color selection (for 24 bit images)
Click on the icon **Color selection**. The dialog box **Color edition** is displayed. This box allows you to select a color by clicking on any part of the Hue and saturation ring (right part of the dialog box) or by entering numerical values for the RGB parameters.
It also allows you to edit a hue. Click on it therefore or enter numerical values for the HSV parameters. The result of your new setting appears in the field **Current color**.

Validate by clicking on **OK**.

Drawing

Select the active color and click on the tool corresponding to the shape you want to draw.

Line
Position the mouse cursor where you want your line to start and click. Drag the mouse until your line is set. Click a last time to define the position of the arrival point.

Rectangle, **Ellipse (circle)**
Place the mouse cursor where you want your shape to start and click. While holding down the left mouse button, drag your cursor until the shape is set. Release the mouse button.

Polyline, **Polygon**
Place the mouse cursor where you want your shape to start and click. Drag the cursor until the first angle position is reached and click then to validate this point. Repeat the last operation as many times as you want to create angles. Click on the right button of the mouse to validate the final shape.

Fill rectangle, **Fill ellipse**
The inside of the created shape is filled with the active color.

Painting

Select the active color and then the desired tool. Four different tools are available:

- **Thin pencil**: the width of the created stroke is 1 pixel.
- **Brush pencil**: the created stroke results of a succession of filled circles.
- **Spray**: this tool allows to create an aerosol painting effect.
- **Fill area**: this tool allows to replace a color by the active one.

Thin pencil, **Brush pencil**, **Spray**
Place the cursor where you want to start painting. While holding the left mouse button, drag the cursor on the area to be painted. Release then the mouse button.

Fill area
Click in the area to be filled with the selected color.

Image deformation

Rotate, resize, etc.
Click on the icon **Rotate, resize,**. Select the operation mode in the dialog box **Image deformation** that is displayed and enter the desired parameters in the corresponding fields.
You can preview any modification by clicking on **Apply** and go back to the first view by clicking on **Reset**.
Validate any modification by clicking on **OK**. The Bitmap Editor is getting closed and the effect is applied to the image.

Resize
This operation allows you to modify the height and the width of the picture. You can either enter the new width and the new height or define the transformation percentages.

Resample
This operation calculates the position and the color of missing pixels. The use of this method allows to reduce the defaults of an image. You can either enter the new width and the new height or define the transformation percentages.

Horizontal and vertical skew
You can either enter the new distance of offset or define the new offset percentage.

Flip and Mirror
Select Flip if you want to apply a centered vertical mirror effect. Select Mirror if you want to apply a centered horizontal mirror effect.

Rotation
Enter an angle between 0° and 360°.

Sharpen and Soften

Both functions are only available with 24 bit images.

Soften
Click on the icon **Soften**. While keeping down the left mouse button, drag the cursor on the area to be softened. Intermediate hues between colors will be created.

Sharpen
Click on the icon **Sharpen**. While keeping down the left button, drag the cursor on the area to be sharpened. Intermediate hues between colors will be suppressed. The contrast is raised.

Effects

These functions are only available with 24 bit images.

Filters
Click on the icon **Filters**. Select the effect you want to apply to the image by using the scroll-down menu of the displayed dialog box. Click on **Apply** to preview the result of the operation. Click on **Reset** to cancel the modification.
Validate the modification by clicking on **OK**.

Embossing (both methods)
Select this option to create a relief effect.

Color pencil effect
Select this option to find the contours.

Soften
Select this option to soften the image by creating intermediate hues between two colors.

Sharpen
Select this option to sharpen the image by suppressing intermediate hues between two colors.

Working with colors

Increasing the number of colors

This operation increases the number of colors up to 16,7 millions. This conversion is very useful to apply the following treatments to the image:

- Effects
- Color reduction
- Soften
- Sharpen

Select **Conversion in 32 bits** in the menu **Conversion** to increase the number of colors.

Decreasing the number of colors

You can only decrease images with 16,7 millions colors.

Color reduction
Click on the icon **Color reduction**. The corresponding dialog box is displayed.
Click on **Fusion** for an automatic fusion. Cancel or validate this operation.
You can select the colors to be treated in the actual palette: while holding down the **Ctrl** key, click on each color you want to select. Click then on **Suppression** to suppress the selected colors or on **Fusion** to merge them.
You can change the parameters of the colors reduction by clicking on **Options…**. Click then on **Color reduction** to apply the reduction.
In case you are not satisfied with the result of your treatment, click on **Reset** to get back the original image.
Once all the parameters are set, click on **OK** to validate.

Gray levels

Conversion in 256 gray-levels
Select **Gray level 8 bits** in the **Conversion** menu in order to reduce the number of colors of an image down to 256 gray-levels.

Gray levels
This operation reduces the number of colors of an image to an equivalent number of gray-levels.

Histogram correction

This option is only available with gray-levels images.

Histogram
Click on the icon **Histogram**. The dialog box **Histogram correction** is displayed.
The graphic displayed in the dialog box represents the way the gray-levels are divided up in the image.
The vertical axe shows the percentage of pixels of which color corresponds to a gray-level quoted on the horizontal axe.

Select the histogram treatment you want to apply. Three are available:

Equalization
This function adjusts the luminosity by dividing up pixels equivalently between different gray-levels.

Specification
This function allows the user to select a bitmap image of which the gray-levels dividing will be applied to the current image.

Stretching
This function adjusts the luminosity so that the histogram contains the whole gray scale.

Choose then between a classical view and a cumulated view.
You can preview the result of your modifications by clicking on **Apply** and cancel it by clicking on **Reset**.

Half-tones

Half tone
Click on this icon to reduce the colors of a black and white image.

Half tone with diffusion
Click on this icon to reduce the colors of a gray-level image and enter the number of gray levels (between 2 and 64). Click on **OK** to validate.

Correction of colors

Correction
Click on the icon **Correction**. The dialog box **Correction of colors** is displayed on the screen. Select one of the pages of the dialog box regarding the kind of correction you want to apply.

RGB Correction (red / green / blue)
A color is reproduced on the screen from the three primary colors which are red, green and blue. For each of these colors you can increase or decrease the parameters (each value is contained between 0 and 255).

HSV Correction (Hue / Saturation / Brightness)
A color can be defined on the screen from three components: hue, saturation and brightness. You can increase or decrease the hue, the saturation (at 255 the image is converted into gray-levels) and the brightness (with 0 for the absolute black and 255 for the absolute white).

Brightness / Contrast
In this page, you can set up the brightness of the image (with +100% for an absolute white and –100% for an absolute black) and increase or decrease the contrast.

Any correction of color can be previewed by clicking on **Apply**. You can get back the original image by clicking on **Reset**.

Working with masks

When it's positioned on the current image, a mask is a closed contour that permits to select a part of the picture.

Using a mask

Polyline mask
Click on the icon **Polyline mask**. Position your cursor on the image and click to mark the starting point of the mask. Drag your cursor until the next point and click to create an angle. Repeat this last operation as many times as there must be angles. Once you created all the angles, click on the right mouse button to validate your mask.

Free drawing mask
Click on the icon **Free drawing mask**. Position your cursor on the image and click. Drag the cursor while holding down the left mouse button. Once you've closed the contour, release the mouse button to validate the mask.

Magic wand mask
Click on the icon **Magic wand mask**. Click on the desired color. Enter the selection parameters in the dialog box that is displayed. Cancel or validate this operation. The chosen color becomes the mask.

Choose a mask
Click on the icon **Choose a mask**. In the displayed dialog box, double-click on the desired kind of mask. Drag your cursor on the image while holding down the left mouse button, and release it once you've reached the desired position and dimensions.

Area of mask becomes the image
Click on this icon. The mask selection becomes the current image.

Show / Hide the mask
Click on this icon to show or hide the mask.

Suppress mask
Click on this icon to delete the mask.

Deforming the mask
The Bitmap Editor allows you to deform a mask. Actually you can move, rotate, align, stretch or project the mask. Click therefore on the desired tool, click then on the mask and drag your cursor while keeping down the left mouse button to deform the mask. Release the left mouse button at the end of the operation.

Copying / Cutting / Pasting
The Bitmap Editor allows you to copy, cut and paste a newly created mask. You can access these functions through the **Edit** menu or through the following shortcuts:

- Copy: **Ctrl** + **C**
- Cut: **Ctrl** + **X**
- Paste: **Ctrl** + **V**

Painting around the mask

Create a mask and select the active color. You can either draw the mask with the brush , fill the area of the mask or fill the area around the mask .

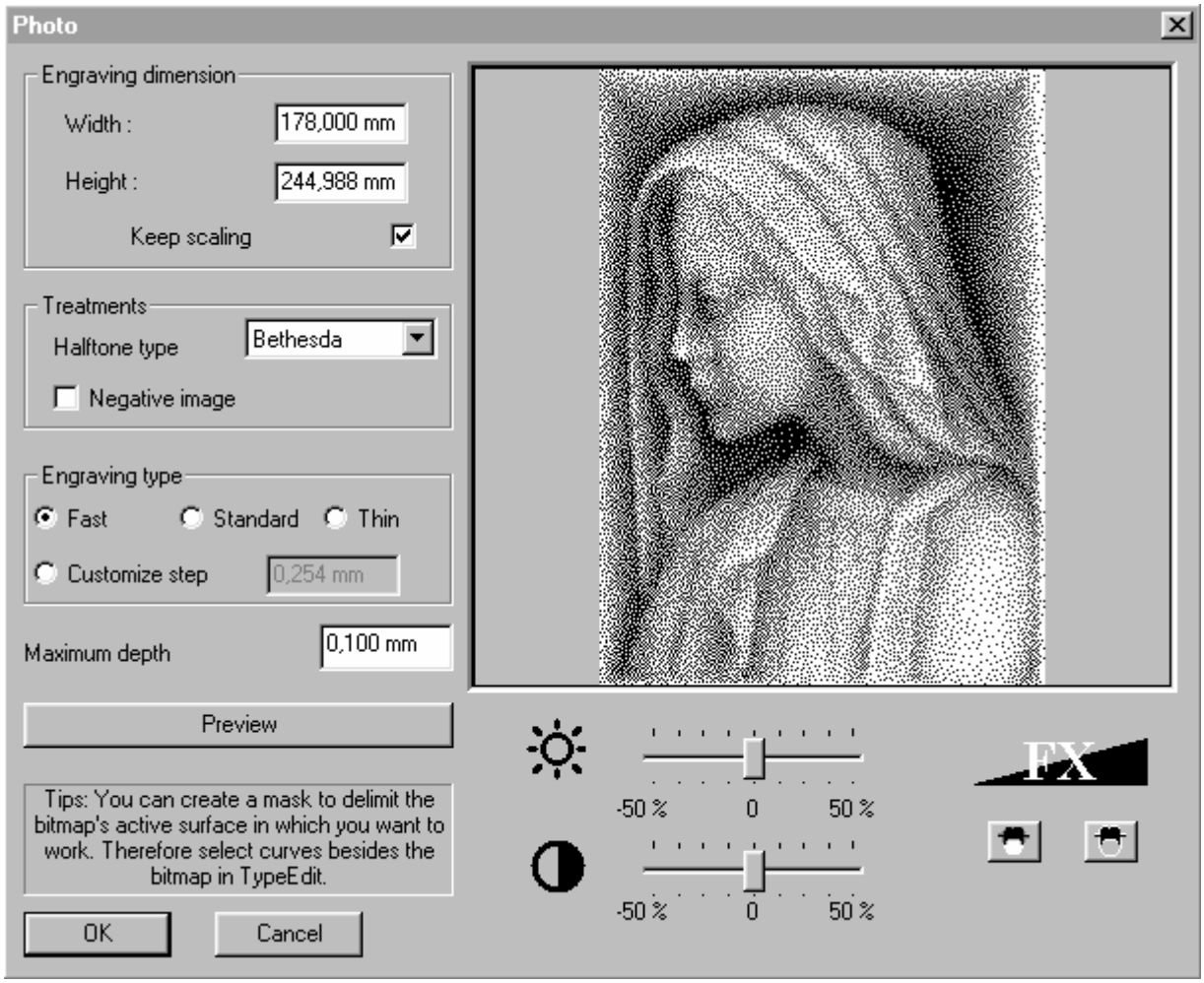
PHOTO

Generating halftone pictures

Photo

This function allows to generate halftone pictures. You can either select a bitmap before clicking on the **Photo** icon or click on the icon before loading a bitmap (you can load it from a scanner).

Once the bitmap is loaded and you clicked on the **Photo** icon, the dialog box is displayed on the screen.

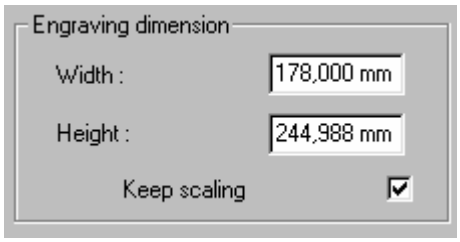


Remark

To display in the right window of the dialog box the results of the parameters setup, click once on the Preview **button.**

Engraving dimensions

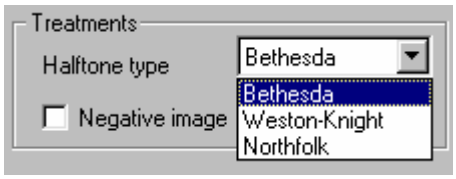
Define the height and the width of your engraving by filling in the corresponding fields as shown below.



Select **Keep scaling** in case you want to modify both dimensions proportionally.

Treatments

Choose among three different halftone types by using the scroll-down menu. You can invert the colors by selecting **Negative image**.

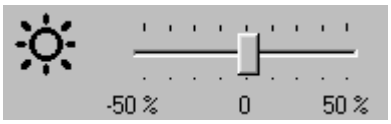


Engraving type

Select the kind of engraving you want to generate. You can also customize its step by filling in the corresponding field.

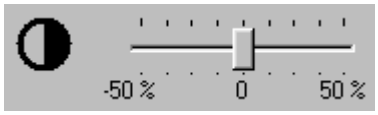
Brightness and contrast

Increase or decrease the brightness of your halftone picture by dragging the cursor shown below.



Don't forget to click on the **Preview** icon to visualize the result of your setup.

Do the same to increase or decrease the contrast of the picture.



Soften

Click on this icon to soften your picture. Click on **Preview** to visualize the result before validating.

Sharpen

Click on this icon to sharpen your picture. Click on **Preview** to visualize the result before validating.

Once the setup is made, click on **OK** to validate. The picture has been turned into a halftone picture.

USING LAYERS

Creating and using a layer or plane

You can place a drawing, or elements of a drawing, in layers (or planes), which are used as tracings piled on a board.

Each layer can be displayed and used individually or with other layers.

There are no limitations neither for the number of layers nor for the number of entities held in each of these layers.

A layer can be a model of construction for the digitalization (See example):

Principles:

- **any layer not in use becomes inactive** (no modifications are therefore possible on the inactive layers),
- **the F3 function** is available for snapping onto geometrical elements of another layer (if it is visible).

A layer is also useful to separate some elements of a drawing into several different layers:

- with the **copy/paste function**, for instance, you can keep all the recessed contours of a complex machining on one layer and all the relief contours on another layer.

Example: Base for the digitalization of a logo

Display a scanned logo, for example. This display is called "layer 1".

In order to register it in a fixed layer which will serve as the basis for reconstruction, you have to create a second layer where the tracing will take place.

Adding / Deleting a layer

Adding a layer

To add a layer, click on the "**+**" symbol located in the lower status bar of the screen.

The status bar confirms that a new layer was created.

You can modify the name of a layer by clicking on the corresponding tab of the status bar. Double-click then on this tab. You can modify now the name of the layer.

Validate the new created name by clicking outside the status bar.

Deleting a layer

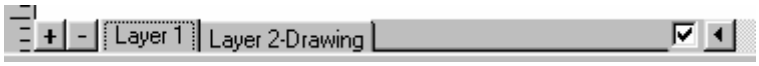
Select the plan you want to delete by clicking on its corresponding tab in the lower status bar. Click then on the - symbol in the same status bar.

Example (deleting the second layer):

In the example shown above, the second layer is selected. Click on "-" to delete it.

Displaying a layer

Click on the current name of the layer found on the status line at the bottom.



Remark

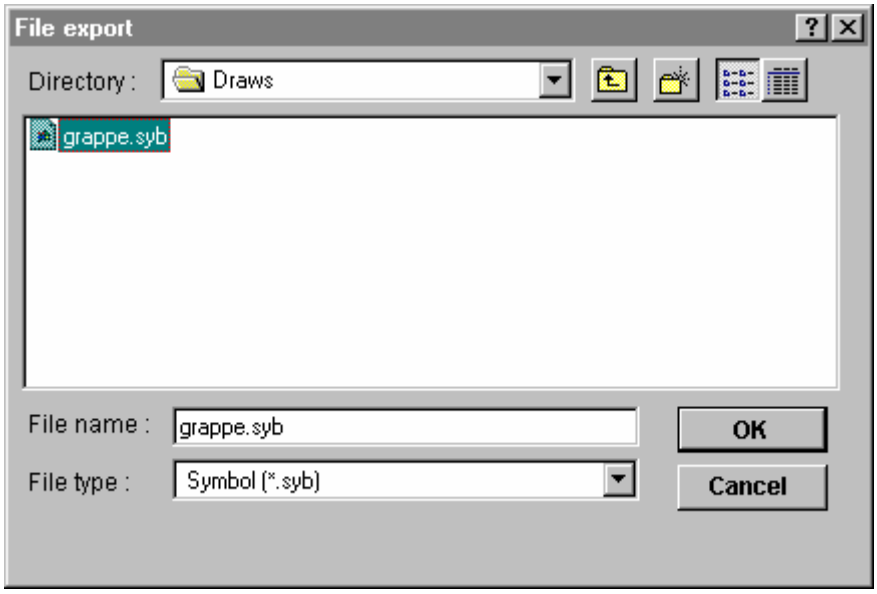
To display only one layer on the screen, unselect the box at the right of the status bar as shown below.

USING SYMBOLS

Creating a symbol

The software includes a symbol library where you can add your own for later use. In that way any repetitive shape or drawing can be kept and is easy to reach.

Select the object to be added to the symbol library and click on **Export** in the **File** menu.



In the dialog box, choose the directory where you want to keep your symbol, then select the **Symbol (*.syb)** file type in the scroll down list. Enter the name of the symbol (8 characters maximum) in the field **File name** and validate.
Example: Grappe.syb. The new symbol now belongs to the library.

Remark

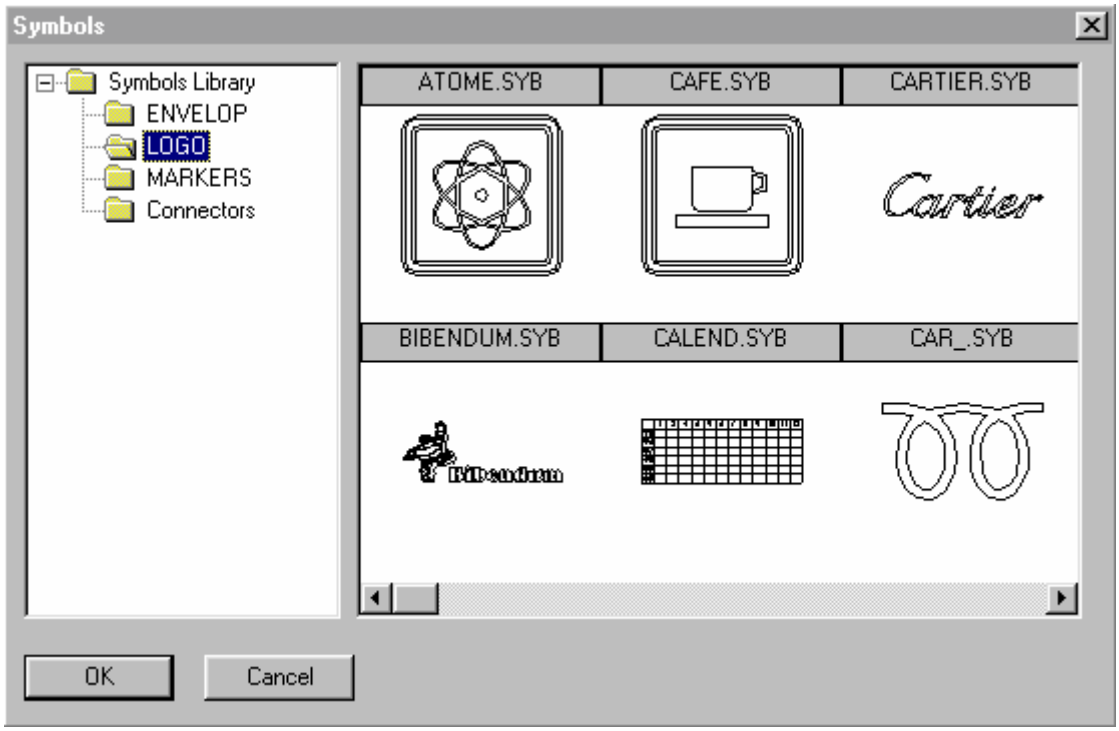
Make sure you are using the .syb **extension so that it will appear afterwards in the symbol library.**

Using the symbol library

Symbols

Click on the **Symbols** icon and then select a symbol from the list provided in the library. In order to help you with the selection, a preview window, which displays the symbol corresponding to the selected name, has been provided on the right hand side.

After selecting the desired symbol, validate by clicking on **OK** and position it on the active surface.



CAM MODULE

STRUCTURE OF THE CAM MODULE

Sending to machining

Click on the icon **CAM module** to transfer the composition to the CAM (machining) module distinguished by its blue screen.

Remark

When one or several elements are selected in the CAD module, your selection is kept in the CAM module.

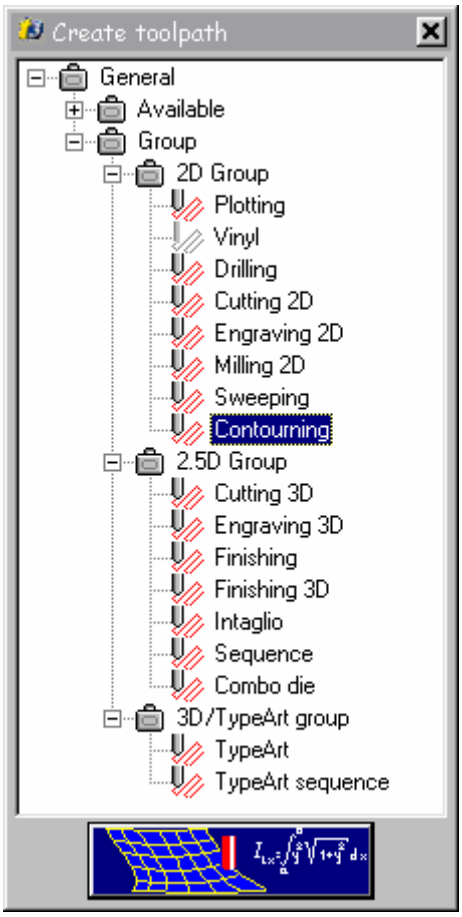
If you forget to save the .vnd file on which you are working, a message reminds you to do so.

To get back in the CAM module, click again on the **CAM module** icon.

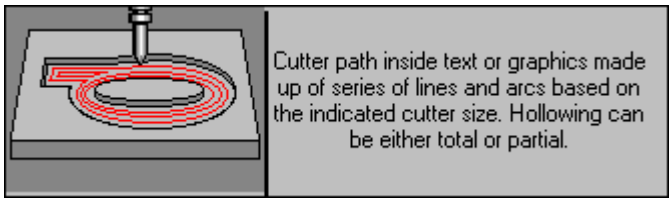
Tools and windows of the CAM module

The CAM toolbar allows you to manage the toolpaths and the tool database.

Click on the icon **Create toolpath** . The dialog box **Create toolpath** gets displayed on the screen. There are two ways to access to the toolpaths. Actually, you can access to any toolpath through two different lists in the dialog box: **Available** and **Group**. The **Available** list gathers all kinds of toolpaths available**,** including your own toolpaths. The **Group** list groups all reference toolpaths in relation to their type (2D, 2.5D, 3D), and cannot be modified.

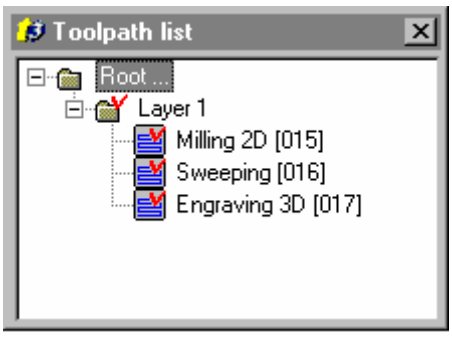


A short explanation concerning the toolpath appears when the cursor remains a few seconds on its name (here for Contouring), e.g.:

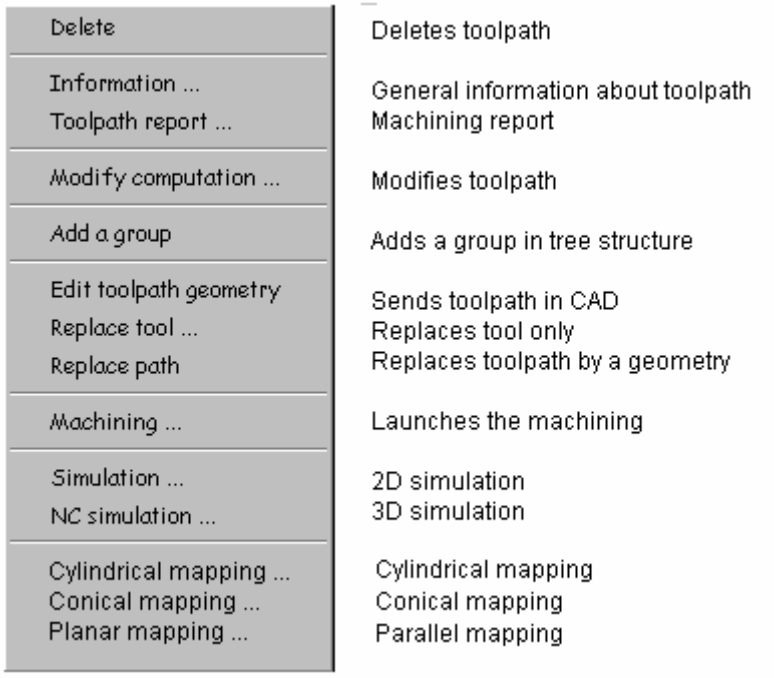

Cutter path inside text or graphics made up of series of lines and arcs based on the indicated cutter size. Hollowing can be either total or partial.

Double-click on the toolpath you want (or select a toolpath and click on the blue icon at the bottom of the window). The corresponding dialogue box opens, with specifying parameters for that toolpath. See the section **Description of toolpaths** below to read how to use the different toolpaths.

Click on the icon **Toolpath list** from the CAM module toolbar. The corresponding dialogue box is displayed on the screen. It contains a list of all the toolpaths you created.



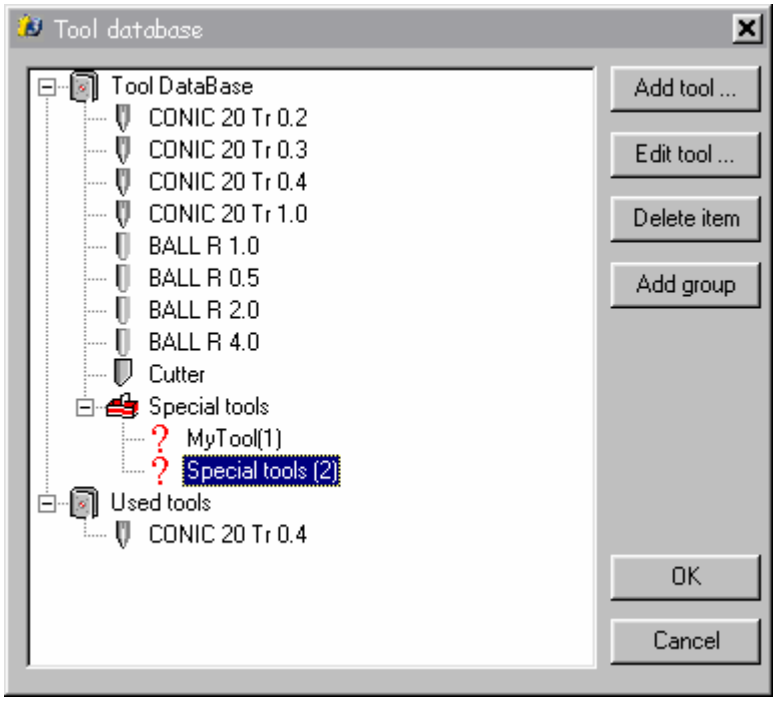
This dialog box allows you to edit toolpaths, to simulate the machining or even to add subgroups using the context menu. To access the context menu, select a layer, a group or a toolpath and click on the mouse right button. The context menu is displayed on the screen.



This dialog box puts at your disposal the **Drag and drop** functions that have become very common in **Windows**. For example, in case you've created several layers, you can select a toolpath from one of the layers, keep pushed the mouse left button, drag it to the folder of another layer and release the mouse button.

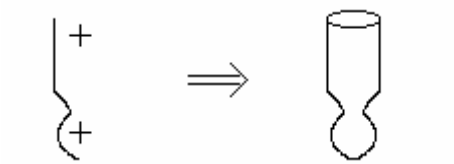
The number between brackets is the extension given to the toolpath file, the base for the name being the name of your document. For example, if you document is called MyLogo, the files MyLogo.001, MyLogo.002 etc. are generated.

When you create a toolpath, you have to choose a tool among those available in the Tool database, by clicking on the icon.



The **Add group** button allows to create a sub-directory in the Tool database, shown by a small red toolbox, in which you can create, delete and modify tools. You can thus group your tools by type of geometry: cones, cylinders, balls, etc.

The **Special tool** icon is very useful for creating a customized tool, based on a form you draw, and of two markers indicating the axis of the tool.
First of all, in the CAD Module draw a profile and 2 aligned markers. Send them to the **CAM module**. Select the profile, then the markers and click on the **Special tool** icon .



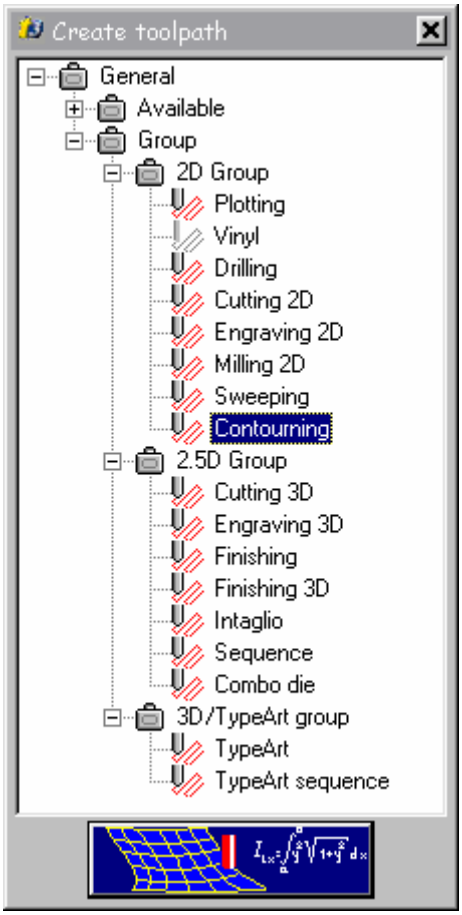
curve (revolution profile) *Generated tool*

You special tool is created, though nothing has occurred on screen. You can see it by clicking on the **Tool Database** icon in which your new tool will appear in a sub-directory called "Special tools", with an icon in shape of a question mark.

The CAM toolbar includes lastly some tools similar to those found in the CAD module, allowing to combine and uncombine contours, and control intersections .

TOOLPATHS: OVERVIEW

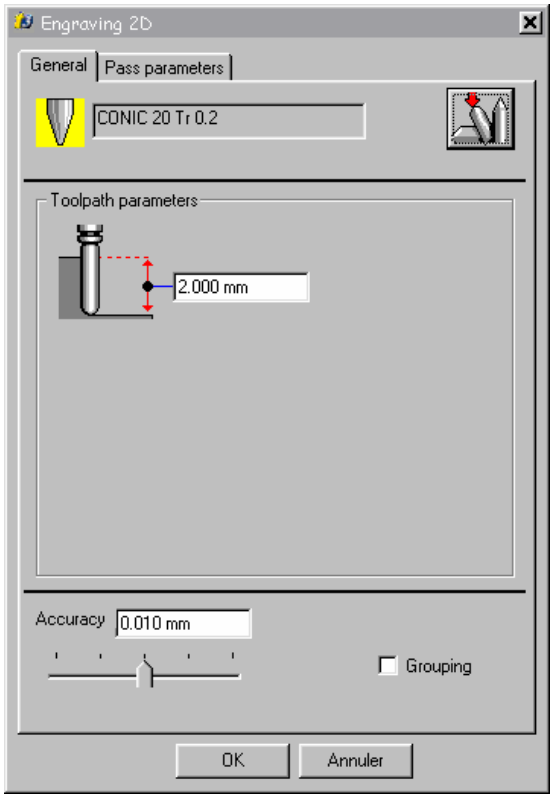
The icon **Create toolpath** displays dialog box **Create toolpath**. You then access to any toolpath through two different lists in the dialog box: **Available** and **Group**.



Each toolpath has some specific parameters, but some parameters are common to many or all toolpaths.

Double-click on the desired toolpath. Enter its parameters in the corresponding dialog box.

General tab



In the upper part of the **General** tab, you must click on the icon to select the appropriate tool in the Tool database. Whatever toolpath you selected, the first information to provide is the tool used for machining.

Depth
Specify the total depth for the milling/cutting.
This information is found also, with more specific parameters, in the **Pass parameters** tab once the tool is chosen.

Accuracy
At the bottom of the dialog box, you can specify an **Accuracy**, which is a tolerance used during the computation of the toolpath. A high accuracy causes longer calculation time, but more precise toolpaths that include a more important number of points.

This parameter can be refined if you notice something unexpected with a toolpath made with the default accuracy (0.01 mm) or for very small toolpaths.

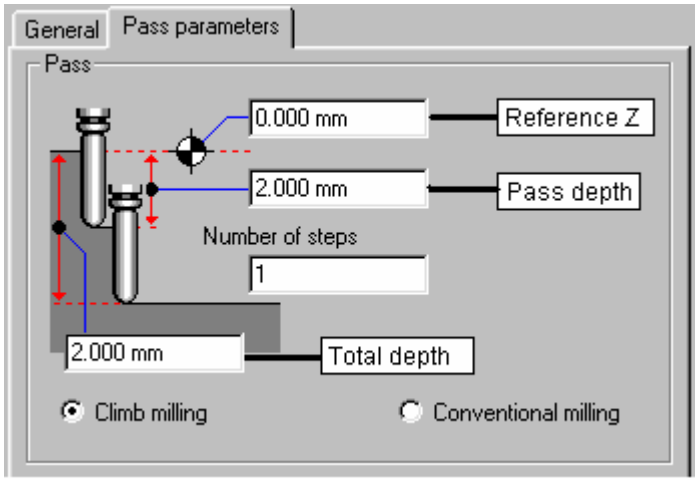
Grouping
The **Grouping** checkbox determine if one or several toolpath files will be generated.

If the box is checked, each selected contour in totally machined, one at a time, creating as many toolpaths as items.
If the box is not checked, the computation is done globally and generates only **one toolpath**, irrespective of the number of selected items.

Example:

without "grouping", one generated file *"grouping", two generated files*

Pass parameters tab



Pass
From top to bottom, you can specify the following parameters:

Reference Z for this toolpath. Useful in particular in the case of a uneven material on which you realize several machining.
Pass depth (automatically adjusted according to the number of steps): similar to the parameter defined in the **General** tab, discussed above.
Number of steps or number of passes (automatically adjusted if necessary).
Total depth of the engraving/cutting (= pass depth if there is only one step).

Remark

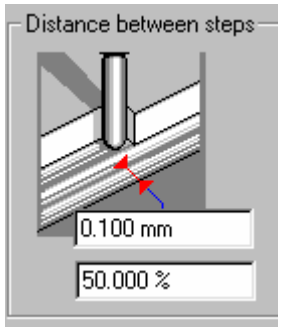
If you use a conical tool, the width of the surface will depend on the depth required.
A simulation display mode would be useful to control collisions.

Climb / conventional milling
Climb milling: The tool is always on the right side of the material:

Conventional milling: The tool is always on the left side of the material:

Distance between steps

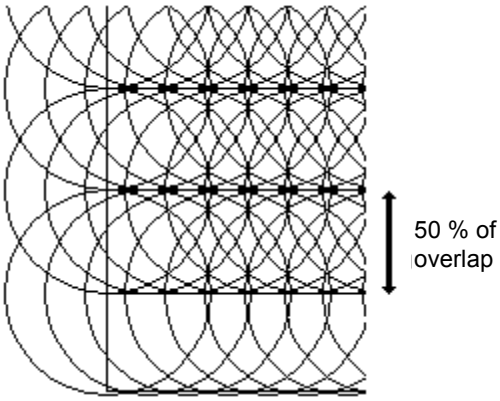
For toolpaths that need several steps (contouring, sweeping, 2D and 3D engraving), the **Distance between steps** is also shown in the **Pass parameters** tab.



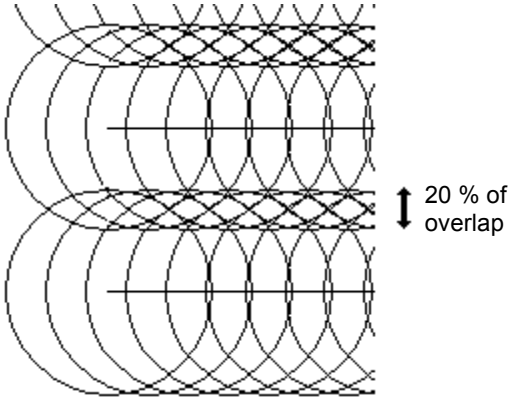
It is the distance between two consecutive steps, measured at the center of the tool. You can directly enter the value (in the unit you have chosen), or a percentage: the other value is automatically deduced.

NB: the percentage is indicated for the diameter of the tool, but according to the shape of the tool, different active parts are considered (spherical toll = radius, flat end tool = tip).
By default the value is 50 %, which is the usual value in order to leave no material between two steps.
The difference between this value and 100 % is the overlapping distance of the pass or **overlap**.

Example



Distance between steps of 50 %



Distance between steps of 80 %

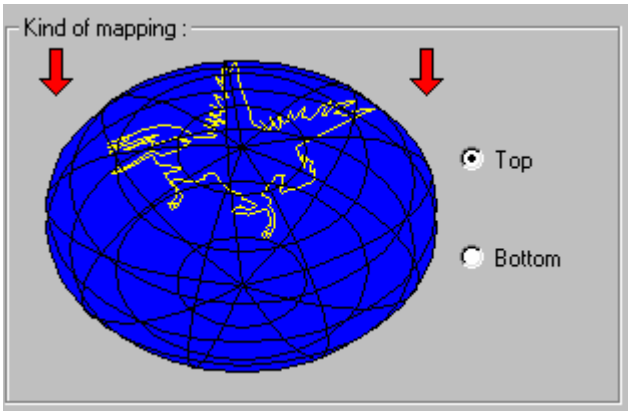
Skip external

For 3D toolpaths using with true angle (3D engraving, finishing, 3D finishing, intaglio), the **Pass parameters** tab includes also a check box called **Skip external frame**. The finishing tool does not engrave the external frame, which saves time and avoids unnecessary usage of the tool.

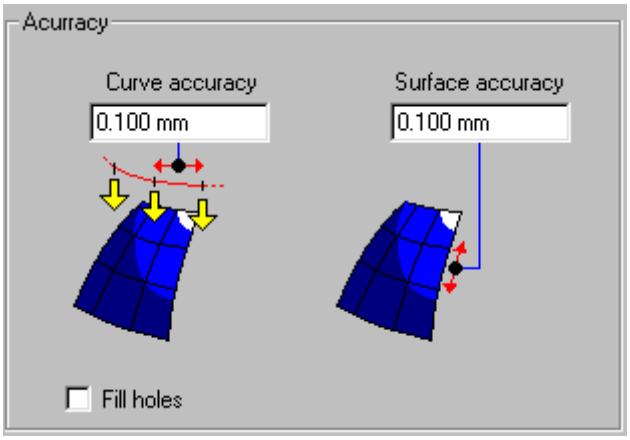
Projection tab

A **Projection** tab appears when a TypeArt object or a surface as well as a curve are included in the items selected for engraving, in order to specify the conditions for projecting the curve on these surfaces.

Specify here the **Accuracy** for segmenting the curve before the projection. If you map on a 3D surface, you must also specify if the projection is done from the top or the bottom of the volume.



In this kind of projection, you should also indicate the accuracy of the segmentation of the surface. The lower the values of **Accuracy**, the more precise the projection (… and the longer the computation time).



According to the nature and shape of the surfaces, some holes can exist or appear. The check box **Fill holes** allows to fill these holes (are considered as "holes" the holes twice as big as the required curve accuracy).

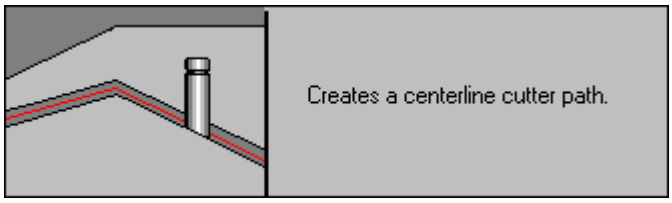
Tool shape

For most toolpaths (Contouring, Intaglio, etc.), the geometry specific to the chosen tool greatly influences the result of the engraving. It sometimes prevails over parameters you enter in the dialog box for your toolpath. Don't forget to make several tests!

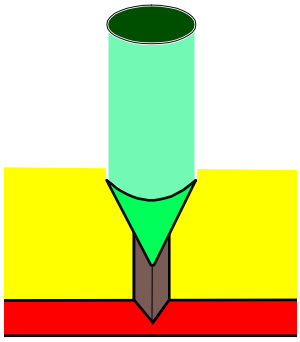
TOOLPATHS OF THE 2D GROUP

Read the overview about toolpaths before reading this specific information for toolpaths used for 2D machining.

Plotting

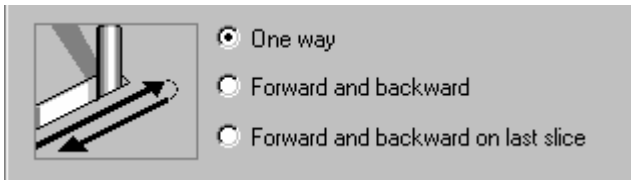


Plotting is the only tool path function available for both open and closed contours.
The function is particularly adapted for the use of single line fonts.
The machine will move exactly like a pen. A line drawing becomes one line on the material being cut or engraved.

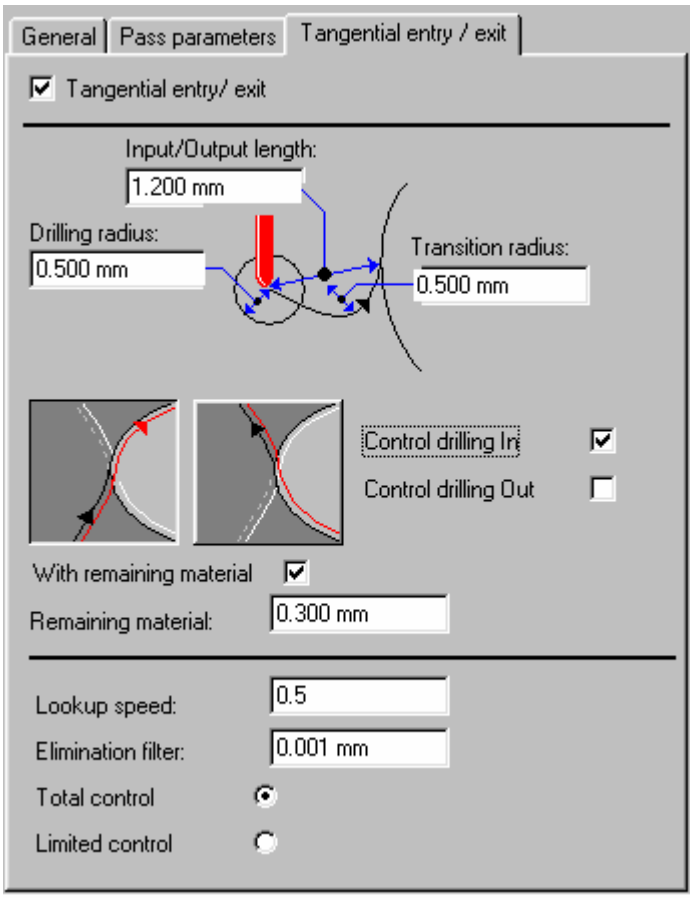


Choose a tool among all tools in the database and enter the depth value.

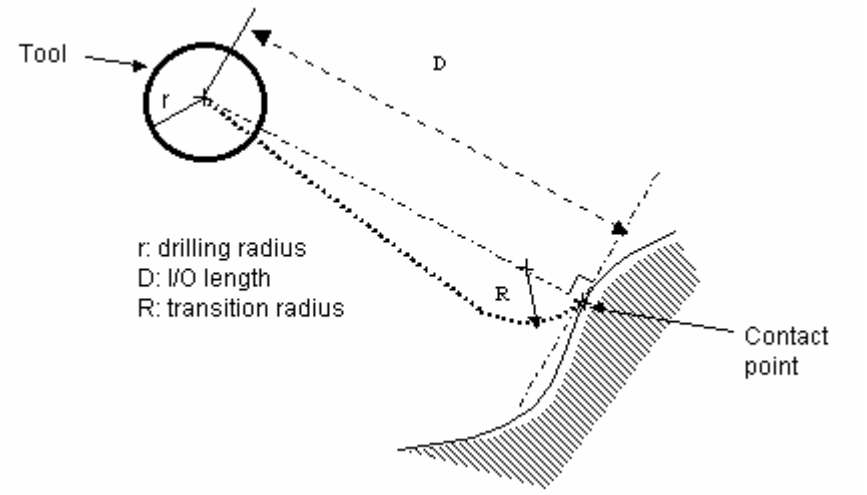
The tool can be asked to go back on the path following the tool center to remove material for a better plotting quality.



"Expert Mode" checkbox
Calls an extra tab to define the tangential entry for the plotting.
This function allows to modify closed contours by adding a tangent entry or exit. Open contours are unchanged.



Check the **Tangential entry/exit** box to enter your parameters.

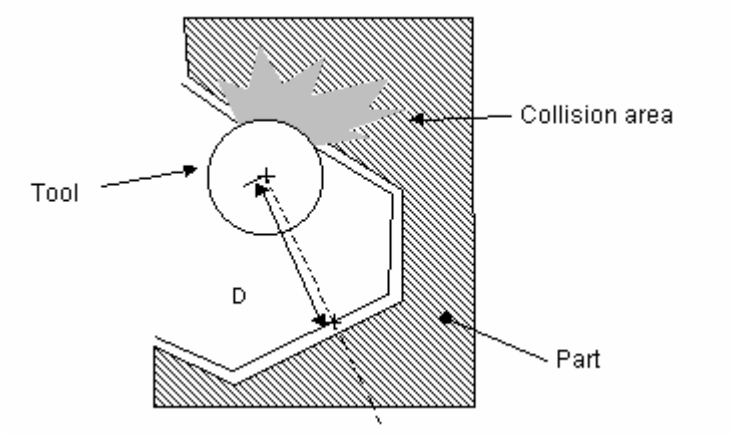


- Drilling radius (r)
The value of the radius at the starting point and/or point of exit created to control an eventual collision on the existing contour(s). This radius must be < D.

- Input / output length (D)
The length between the starting point and the point of contact on the contour.
Whatever the radius of the tool, D is never above 2R.

- Transition radius (R)
The input / output radius.

Control drilling in
This option makes sure that no element of the contour is closed to the starting point than the value of the radius.



Control drilling out
Similar, but for exiting the contour.

With remaining material
If this option is validated, starting and finishing points will be at the distance entered in the **Remaining material** field. If the option is not chosen, the points will be at the same location.

- Remaining material
The length removed from both sides of the contact point (to prevent the piece from falling).

Lookup speed (0.05 to 0.95)
This function corresponds to the accuracy of the search for another starting point along the contour in case of collision. The smaller the value, the longer the search.

Precision filter (0.001)
If the distance of collision is less than or equal to this distance, it is assumed that there is no collision. (Do not change the default value).

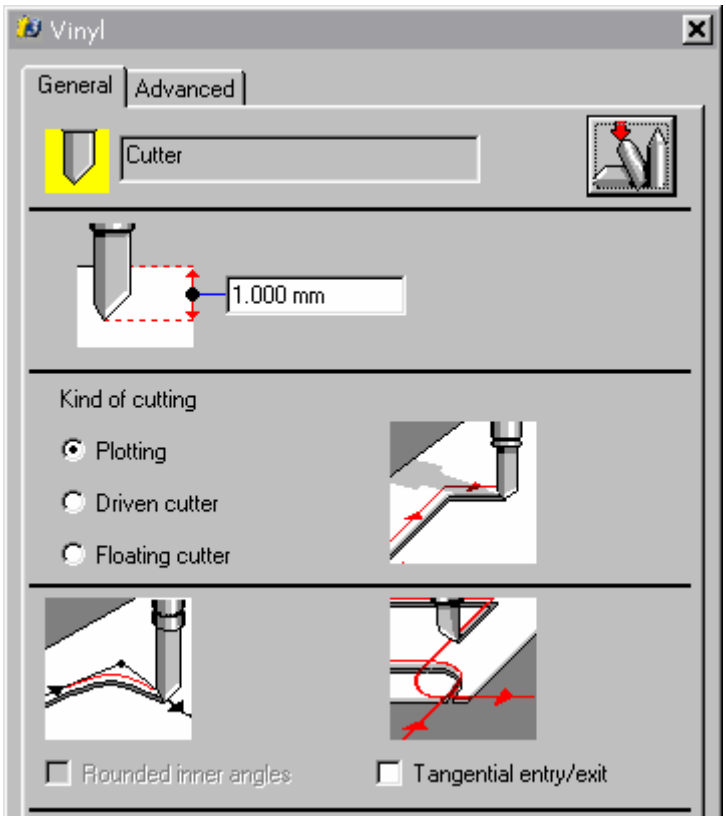
Total control: crash control is done on the contour and on the neighboring contours.

Limited control: crash control is done on the contour itself.

Vinyl

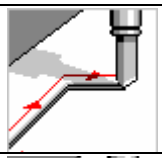
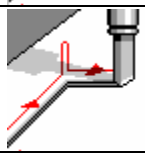
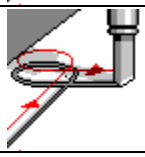
Cutting toolpath. You must first define your tool.

- **General tab**



Specify first the cutting depth for the tool.

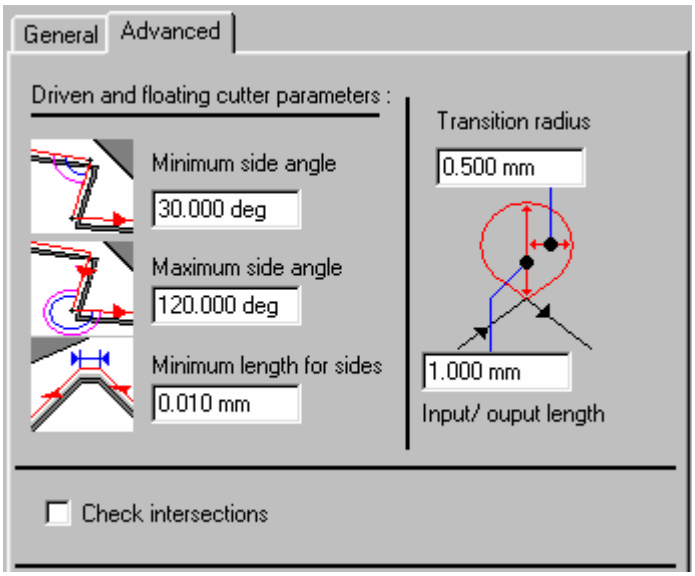
Section **Kind of cutting**

	**Plotting**: the cutter follows only the contour.
	**Driven cutter**: the cutter follows only the contour. When it reaches an angle, the cutter moves up, changes orientation and enters again into the material. The range of angles for which the cutter goes up is set in the advanced parameters.
	**Floating cutter**: the mobile cutter follows the contour (in a tangent way). When it reaches an acute angle, it makes an external curve tangent to the trajectory in order to be properly positioned for following the cutting. The range of angles for which the cutter does a curve is set in the advanced parameters.

Rounded inner angles: non implemented.

Tangential entry/exit: the entry and the exit of the tool are done externally to the contour, in a tangent way, which produces a result of a better quality.

- **Advanced tab**



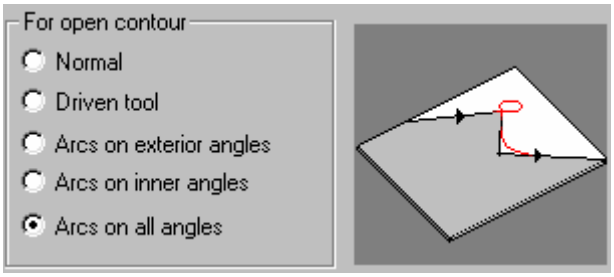
Driven and floating cutter parameters:

Minimum/Maximum side angle: for any angle contained between the chosen values, the tool does a curve (floating tool) or a move up and positioning (driven tool).

Minimum length for sides: below the chosen value the tool does a tangent move.

Transition radius and **Input/output length:**
Only applied to the floating cutter, these parameters allow to set the size of the curve done at each angle (radius and distance input/output).

Check intersections:
When checked, allows to verify that the curves done by the floating tool do no interfere with the original drawing. In such a case, the size of the curves and their orientation are automatically adapted to avoid collisions between the toolpath and the geometry to cut.

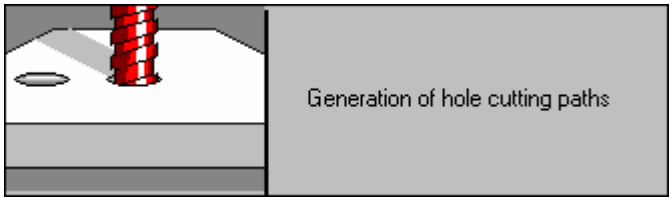


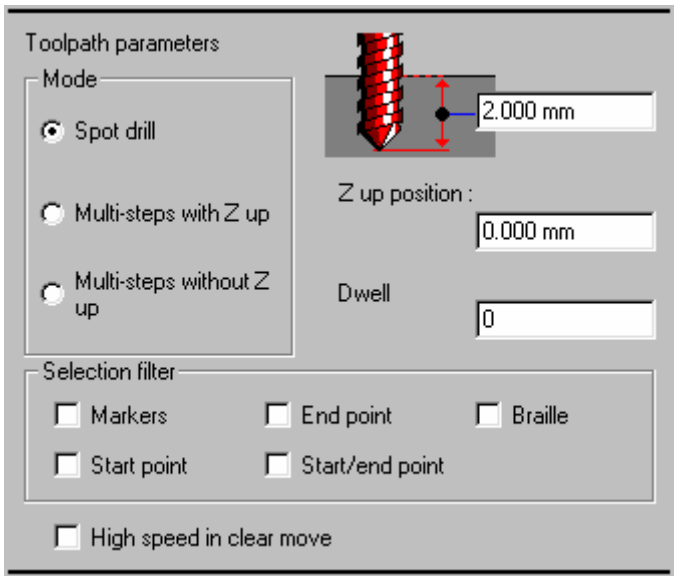
For open contours:

These options are used only for the floating tool and determine the behavior when changing direction.

Normal: no curve is used for the tool. The cutting is done in plotting mode.
Driven tool: the tool moves up at each angle, then enters back into the material.
Arcs on exterior angles: a positioning curve is done for each exterior angle.
Arcs on inner angles: a positioning curve is done for each interior angle.
Arcs on all angles: *c*ombines the 2 previous options.

Drilling or Boring





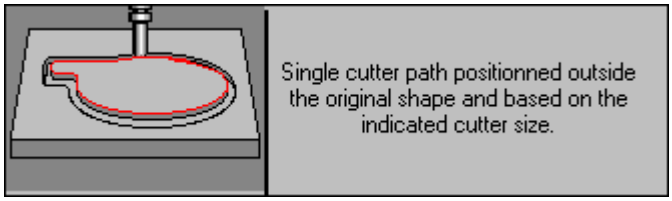
Mode
Spot drill: drilling is made in one step.
Multi-step with Z up: drilling is made in several steps with a move in Z reference position between each step. Is used in particular to evacuate material.
Multi-step without Z up: drilling is made in several steps with an optional move in the Z position specified by **Z up position.**
Drilling depth: drilling depth
Z up position: Z coordinates for movements outside the material.
Dwell: when the tool is at the bottom, it can be useful to add a short waiting time before moving up.

Selection filter
Allows to indicate which elements among the selected contours will be the object of the drilling. The points defined by the Draw holes tool are considered as special markers.
Markers: a drill is made at the position of each selected marker.
Start point: a drill is made at the position of each selected curve starting point.
End point: a drill is made at the position of each selected curve ending point.
Start/end point: combines both previous choices, to take into account closed contours.
Braille: a drill is made at the center of each closed contour.

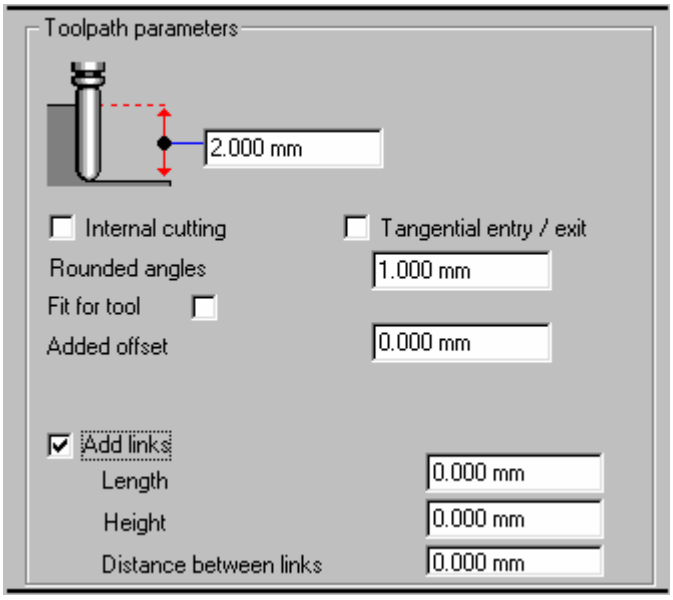
High speed in clear move: by default, the move up is done in working speed. By checking this box, the fast speed defined in the **Technologies** tab will be used.

Remark:
No toolpath is generated for the drilling; visualization must be done in 2D or 3D.

2D cutting



The tool will cut outside the first contour of the drawing (no true angles).



Internal cutting: by default, the cutting is internal to the contour.
Rounded angles: the angles are always rounded, at the minimum taking into account the trajectory of the tool. If a positive value is given for this parameter, it adds another rounding, as if the angle of the contour was a true rounded angle.
Fit for tool: if checked, the rounding is also adjusted to the shape of the tool.
Added offset: the toolpath is moved with the specified value, relatively to the contour. For an external cutting, a positive value will add an offset inwards. For an internal cutting, a positive value will add an offset outwards.

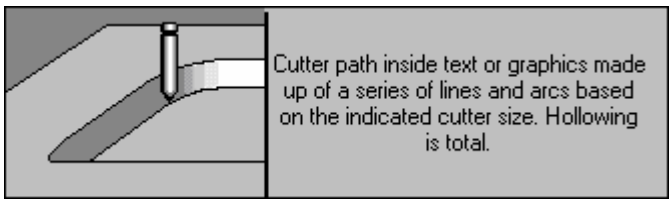
Tangential entry/exit: if checked, the **Tangential entry/exit** tab described above for the Plotting toolpath appears.

Add links
Check this box to avoid that a total cutting make the part bend and maybe warp the cutting. The toolpath does not make a full cutting, but leaves some links for which you specify the **Length**, **Height** and **Distance** between links.

Remark

To cut with a sharp angle from the bottom, activate the option True angle at the bottom **in the** Options **dialog box (tab** Calculation parameters**). Hit the** F10 **key from CAD module to open this dialog box.**

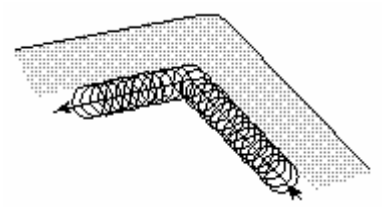
2D engraving


Cutter path inside text or graphics made up of a series of lines and arcs based on the indicated cutter size. Hollowing is total.

This automatic function makes a 2D tool path with complete area clearance.

The cutting width is the distance between two passes on the horizontal plane. It is automatically calculated relative to half the diameter of the chosen tool. You can easily change the value in the **General** tab.

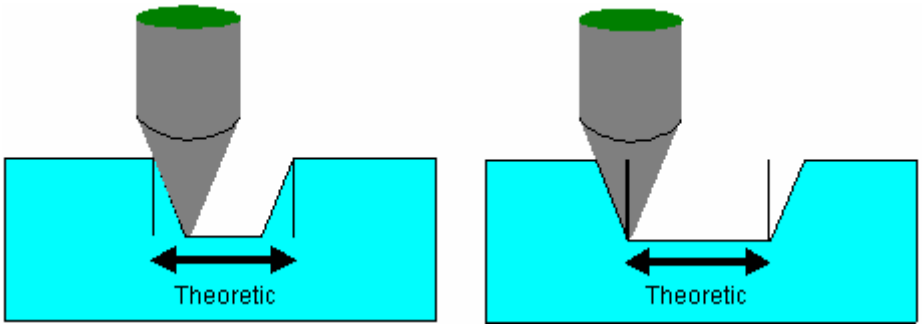
Number of steps
Number of passes to reach the total depth (the depth of the first pass is automatically updated).



One depth
Maximum depth in Z for one pass (number of passes is automatically updated).

2D on surface / 2D on bottom
You chose if the 2D engraving path executed by the tool reproduces the theoretical contour of the drawing on the top or on the bottom of the engraving (specially useful when using conical tools).

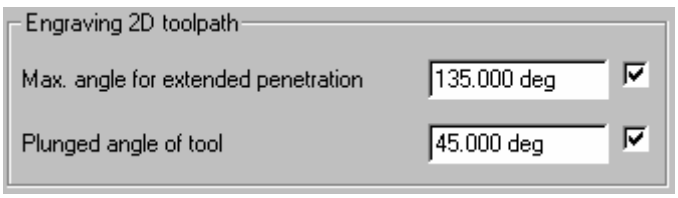
Example



2D on surface *2D on bottom*

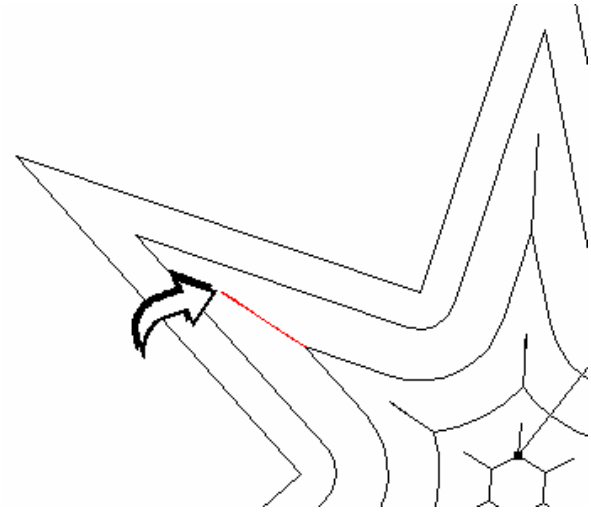
Distance between steps
Quantity of material removed by concentric passes (see the overview).

F10 - Specific parameters
Two items in the Specific parameters tab (Options - F10) are related to the work on angles for engraving.



Max. angle for extended penetration
In the case of acute angles, the tool advances into each angle. This prolongation of the path prevents from specifying narrower steps (or from leaving empty areas).

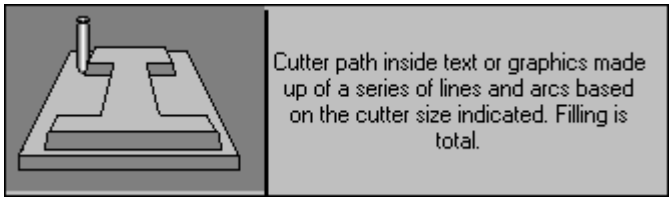
Example :

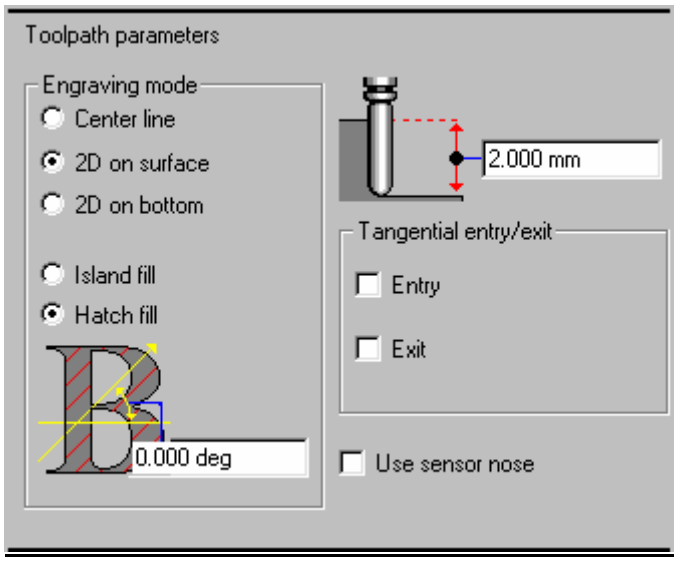


No angle cutting above the given value (default angle = 135°, max. = 180°).

Plunged angle for tool
Specify the maximum angle for tool plunging (from 1° to 89°).

2D Milling


Cutter path inside text or graphics made up of a series of lines and arcs based on the cutter size indicated. Filling is total.



Engraving mode

Center line

The centerline filling in is the same as the toolpath of the Plotting mode (see this section).

2D Milling

The milling of an engraving in this mode is executed by successive passes (hatching or contouring).
Two fillings in styles are available:

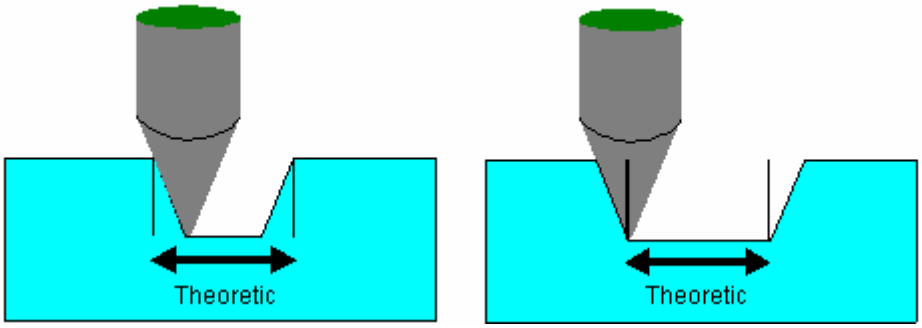
2D on surface

The 2D engraving path executed by the tool reproduces the theoretical contour of the drawing on the surface. The closed contours are milled first, then the opened contours.

2D on bottom

The 2D engraving path executed by the tool reproduces the theoretical contour of the drawing on the bottom of the engraving.

Example



2D on surface *2D on bottom*

Hatching angle

It can be used either to obtain a visual effect on the bottom of the engraving or to minimize the size of the files (by using the main axis of the drawing as the hatching axis, it can minimize the clearance for instance).

Use sensor nose

Select this option if you are using a flatness sensor, the spiral area clearance will be automatically set.
You will avoid the tool from dropping into areas already milled.

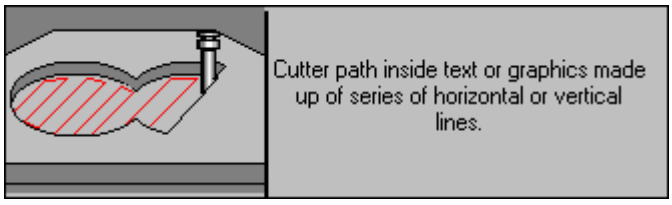
Remark

Once the Use sensor nose **option activated, you can no longer define the hatching angle, since milling is executed by concentric passes from the external to the internal contour.**

Tangential entry / exit

Use this function if you wish to start or to end your work with a displacement tangent to the contour.

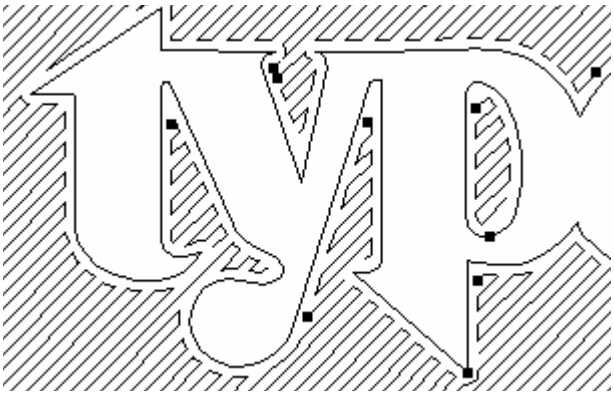
Sweeping or Parallel hatching



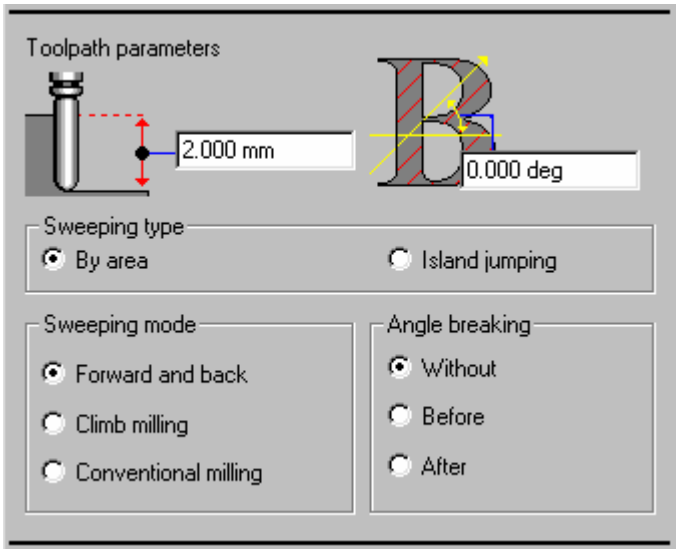
The sweeping function allows you to program the removal of material by parallel motions according to the chosen angle. The function thereby minimizes the number of times the tool has to ascend and to descend.

Sweeping is very useful for enlarging the engraving quickly and can be executed with a large tool either

cylindrical or conical. By choosing various sweeping angles, you can create different optical effects on the bottom of the engraving, or minimize the size of the machining files (by following the main axis of the drawing, the number of ascents are minimized).

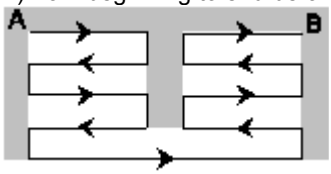


45°Angle

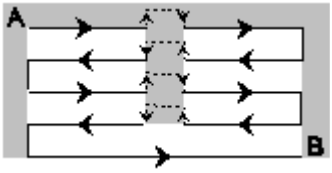


Sweeping type

By area
The tool machines the whole zone (zone A) from beginning to end before starting on another zone (zone B).

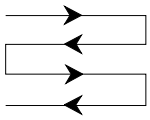


Island Jumping
The tool works from top to bottom jumping over the “islands” (non-machined areas).



Sweeping mode

Forward and back
The tool follows its path continuously without lifting and always inside the material.

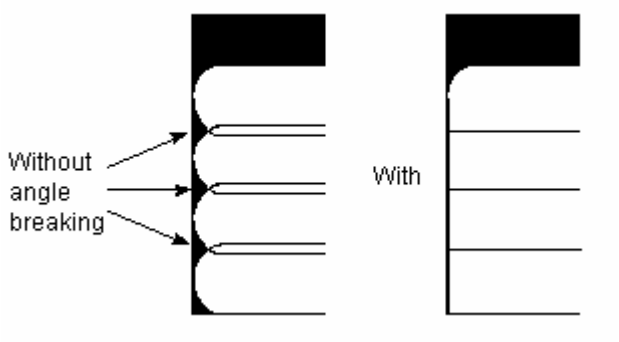


Climb milling
The tool is always on the right side of the material:

Conventional milling
The tool is always on the left side of the material:

Angle breaking

Trimming before or after sweeping.



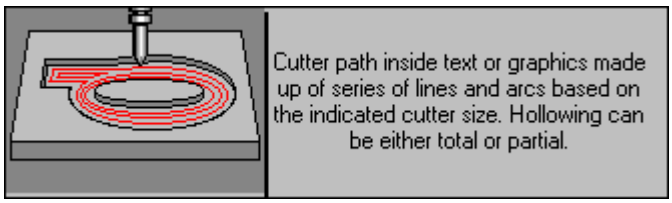
Without
No angle breaking is done.

Before
The tool will first trim the material and then go into the sweeping mode.

After
The trimming is done after the sweeping is finished.

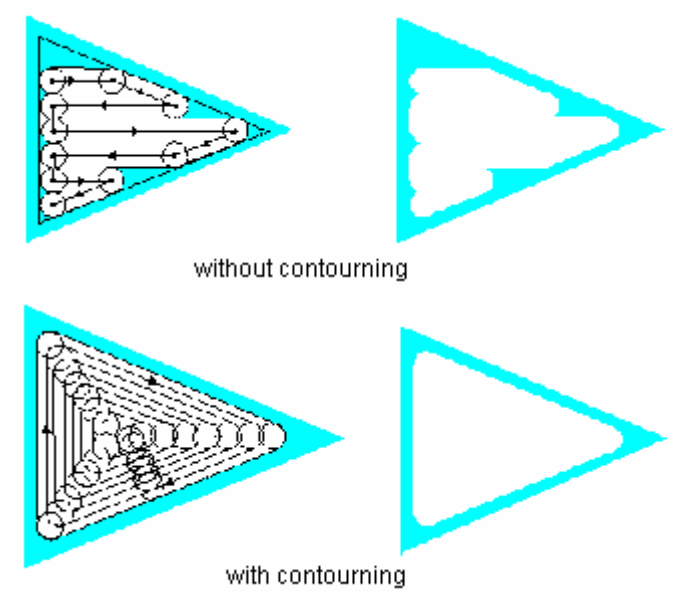
Note
The indication "Climb Milling / Conventional Milling" of the Pass Parameters **tab applies to the angle breaking pass. The indication "Climb Milling / Conventional Milling" of the** General **tab applies only to the sweeping mode itself.**

Contouring or Spiral area clean


Cutter path inside text or graphics made up of series of lines and arcs based on the indicated cutter size. Hollowing can be either total or partial.

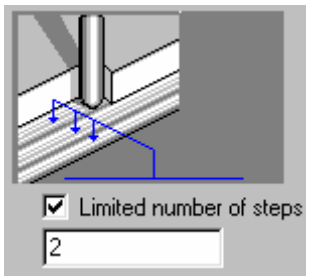
This function allows you to program the removal material through concentric motions by progressively approaching the original form. The tool shape has such an influence on the machining, that if the shape is not appropriate for engraving the theoretical contour, no toolpath will appear on the screen.

See the general parameter in the **Overview** at the beginning of the chapter.



By default, all the material will be removed (total recess).

Total spiral cleaning



Only the number of the desired spiral passes will be executed by the tool (limited recess).

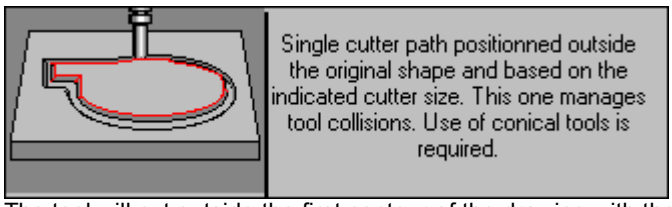
Spiral cleaning limited to 2 passes

TOOLPATHS OF THE 2.5D GROUP

Toolpaths in 2.5 dimensions concern paths that can manage tool collisions and true angles. The use of the term "3 dimensions" are justified for some people as true angles use a third axis. This axis being often considered as an half-angle, other people will prefer the term of "2.5 dimensions".

Read the overview about toolpaths before reading this specific information for toolpaths used for 2.5D machining.

3D cutting


Single cutter path positionned outside the original shape and based on the indicated cutter size. This one manages tool collisions. Use of conical tools is required.

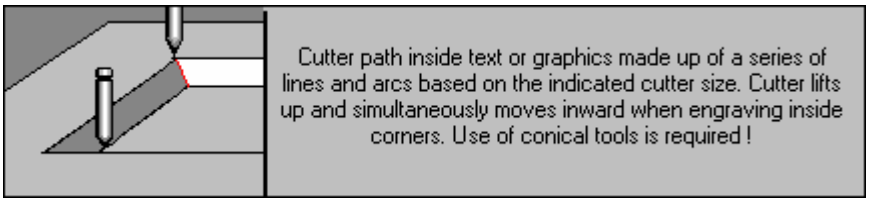
The tool will cut outside the first contour of the drawing with the true angle at the top.

See the section 2D Cutting, and the general parameter in the **Overview** at the beginning of the chapter.

Remark

To cut with a sharp angle from the bottom, activate the option True angle at the bottom **in the dialog box** Options **(tab** Calculation parameters**). Hit the** F10 **key in the CAD module to open this dialog box.**

3D engraving

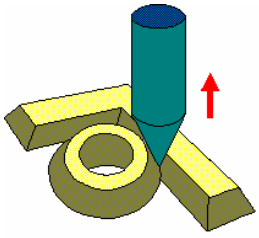

Cutter path inside text or graphics made up of a series of lines and arcs based on the indicated cutter size. Cutter lifts up and simultaneously moves inward when engraving inside corners. Use of conical tools is required !

This automatic function makes a 3D tool path (the area clearance and finishing are carried out with the same tool.

See the section 2D engraving, and the general parameter in the **Overview** at the beginning of the chapter.

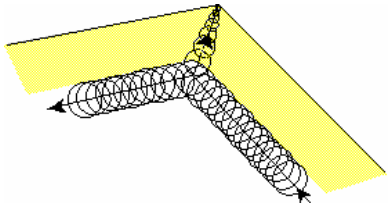
Two cases exist when the tool path calculation is in 3D:

- to avoid tool collisions (between 2 letters in relief for instance)

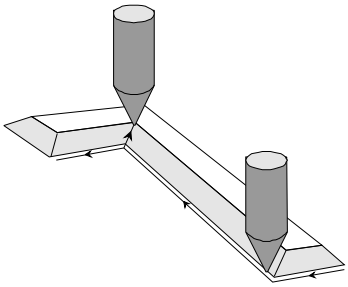


- when creating sharp angles: Inside each angle of a drawing, the tool ascends in order to improve the

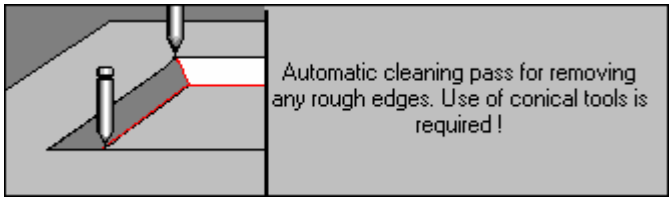
detail of this angle with sharp lines.
This function is called “true angle at the top”.



The same function exists to improve details of the angle at the bottom, i.e. at the bottom of the engraving. This function is optional and can be activated in the Options window (Calculation parameters tab**).**

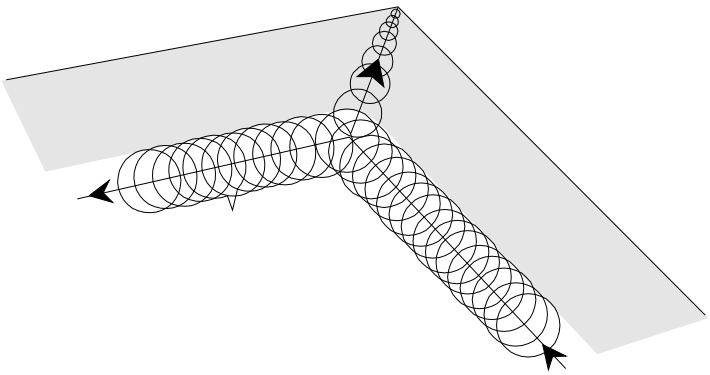


Finishing



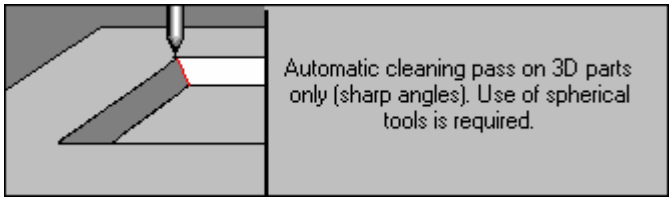
Finishing, with its sharp angles, is the last phase of the engraving process. The tool will follow the outline, or contour, of the design, avoid collisions, and ascend at each sunken angle like a relief.

See the general parameter in the **Overview** at the beginning of the chapter.



Sharp corner
The tool rises within each angle.

3D finishing

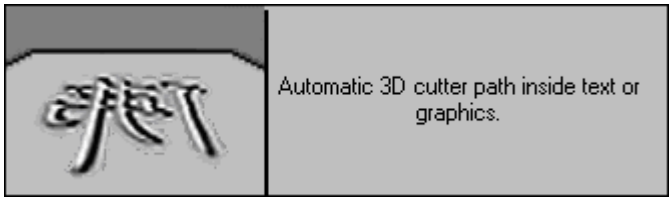


This function calculates only the tool path in 3D areas and take into account sharp angles and tool collisions.

With this function, use only small tools and only in 3D areas. The function allows you not to wear out small tools.

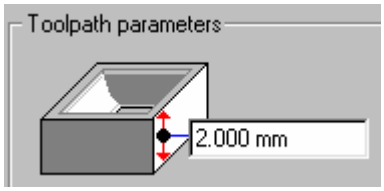
See the general parameter in the **Overview** at the beginning of the chapter.

Intaglio



This automatic function allows you in a single operation to make exceptionally beautiful 3D engravings in recess. It is especially adapted for 3D engravings in wood or Plexiglas.

The depth of your engraving is automatically calculated and takes into consideration the width of the path, or traces, to be obtained. For that reason you must give the thickness of the material after having chosen the appropriate tool (not the depth of engraving).



Material thickness
In this function you do not need to indicate the depth of the pass since it will change and be dependent on the drawing, but you do need to indicate the thickness of the material i.e. the thickness that should not be exceeded.

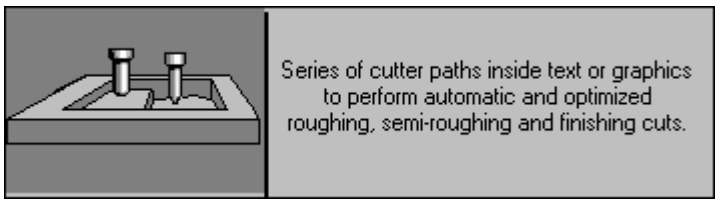
If the depth given to the tool exceeds the material thickness, a message will appear on your screen. In this case, choose a thicker material or a larger tool.

Remark

It is obvious that only conical tools can be used.

F10 - Specific parameters
One item in the Specific parameters tab (Options - F10) is related to the maximum angle above which the tool does not make sharp angles.

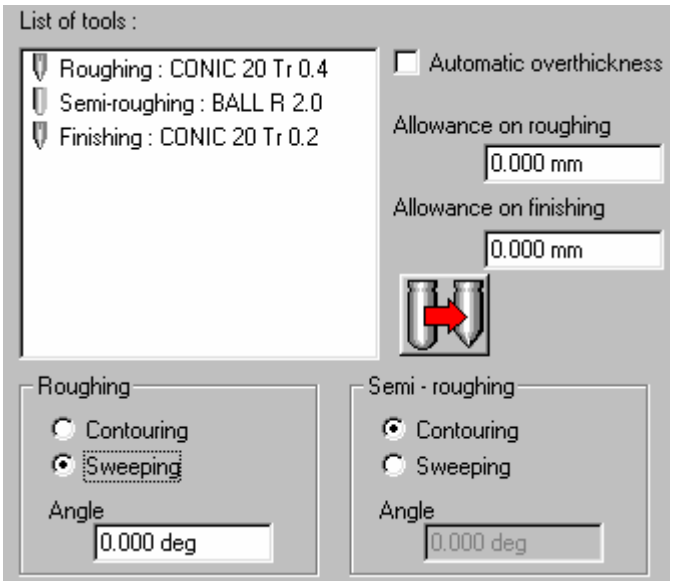
Sequence


Series of cutter paths inside text or graphics
to perform automatic and optimized
roughing, semi-roughing and finishing cuts.

This function allows you to control the last 3 or more tools of a 3D engraving job. Before activating it, you have to complete the roughing of the project with Sweeping or Contouring.

The 3 tools to be defined are the following:
- the roughing tool,
- the semi roughing tool,
- the finishing tool.

The roughing tool works in 2D. It carries out the first step, usually not very close to the theoretical contour.
The semi roughing tool works in 2D. It will clean the remaining imperfections of the material.
The finishing tool works in 3D and will finish the engraving by passing the closest to the theoretical drawing.



For each toolpath, you must select the proper tool by double-clicking on the button. A new tab is added to the dialog box, allowing you to enter parameters for this tool. The **Sequence** tab is used for general parameters.

Overthickness
The overthickness is the material left at one step for next steps. An automatic computation (10% of the finishing tool tip) is done if the **Automatic overthickness** box is checked. Otherwise you can specify the overthickness individually in the following fields.
Allowance on roughing
Overthickness required for the last roughing step.
Allowance on finishing
Overthickness demanded for the finishing step. The semi-roughing tool will go in all areas between these two allowances.

For the roughing and semi-roughing, you indicate whether it is a **contouring** or a **sweeping** (and specify its angle).

Remark 1

The diameter of the finishing tool's tip should not be less than half of the semi roughing tool's tip for an

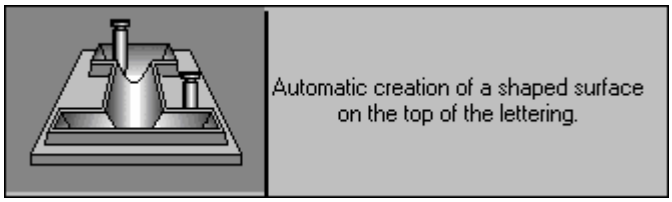
optimal efficiency.

Example: *Semi roughing tool* *Tip diameter of 0,3*
Finishing tool *Minimum tip diameter of 0,15*

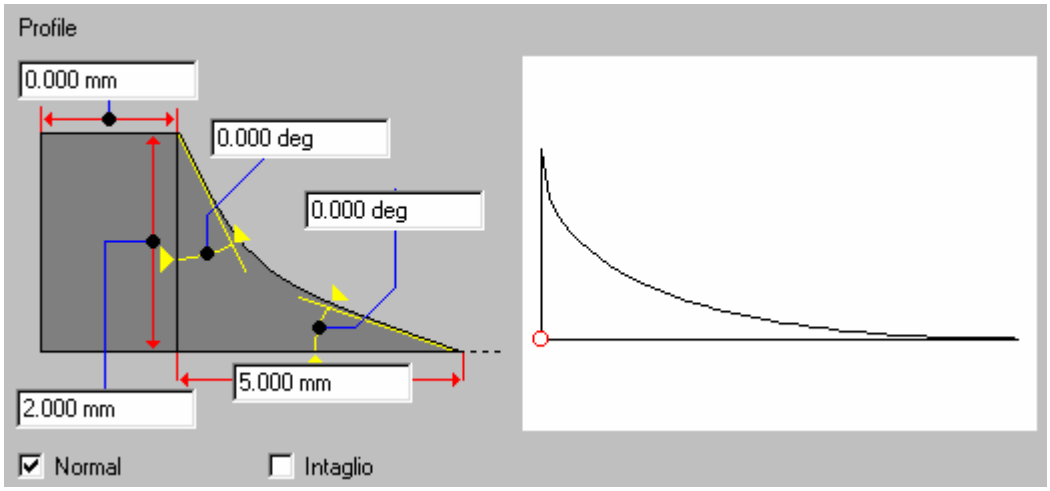
Remark 2

Automatic recessing is particularly efficient if the engraving depth is the same for all the selected tools.

Combo die



This function allows to make section in 2D engraving with several concentric passes.



You define values for the half-profile of the section, by entering the following values (from top to bottom):
Width on the edge: width of the material left on the edge.
Starting angle, **Ending angle** for the profile: both values can be specified in degrees, or deduced by moving the red point in the right pane.
Maximum width of the profile: maximum propagation of the section (for the half-profile).
Total depth of the profile.

Normal: by default, includes several concentric passes.
Intaglio: extra pass (3D) to clean the irregular parts of the section, for example true angles.
At least one of these boxes should be checked.

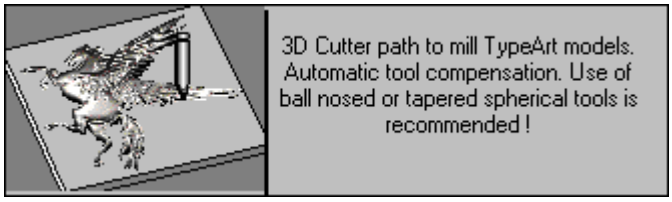
By checking the **Automatic sequence** box, a **Sequence** tab appears, allowing to enter parameters for the sequence (see this toolpath).

TOOLPATHS OF THE 3D / TYPEART GROUP

The toolpaths of this group support 3D objects.

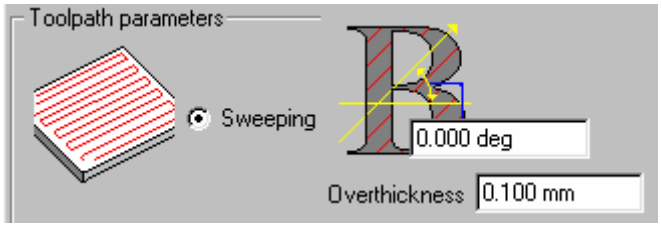
Read the overview about toolpaths before reading this specific information for toolpaths used for 3D machining of TypeArt objects.

TypeArt

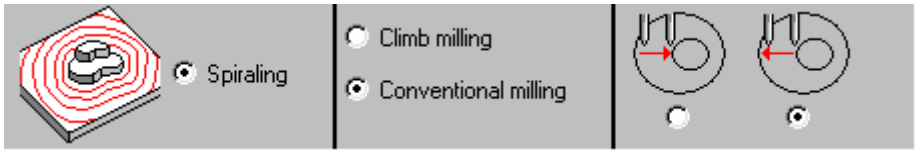

3D Cutter path to mill TypeArt models. Automatic tool compensation. Use of ball nosed or tapered spherical tools is recommended !

The TypeArt toolpath allows the machining of 3D objects created with TypeArt.

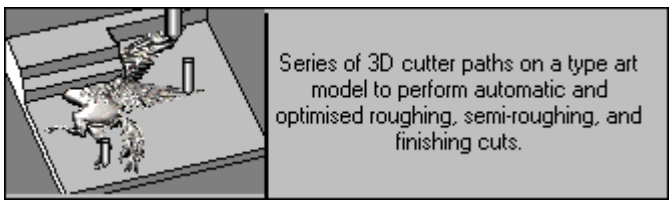
Sweeping mode
Choosing this mode, you can specify two parameters: the **sweeping angle** and the **Overthickness** (or left material).



Spiraling mode
With this mode, you can specify two parameters (in addition to the **Overthickness):** the **milling direction** (see the explanations given for the Sweeping) and the **machining direction** (from inside out, or outside in).



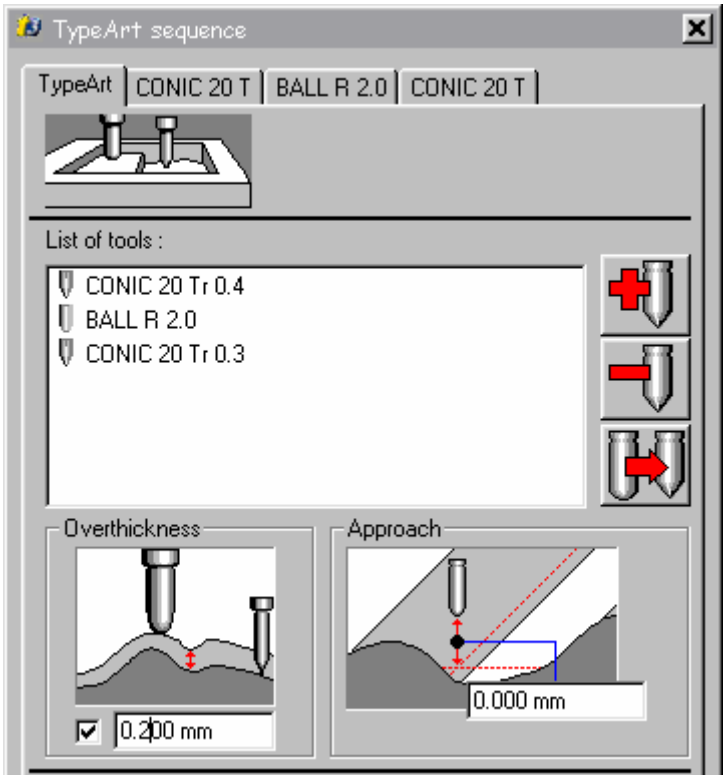
TypeArt sequence


Series of 3D cutter paths on a type art model to perform automatic and optimised roughing, semi-roughing, and finishing cuts.

This function is an automatic sequence for machining 3D objects created with TypeArt.

You can see below a **General** tab with a sequence made of:

- A roughing (with a conic tool),
- A semi-roughing (with a rounded tool),
- A finishing (with a conic tool).



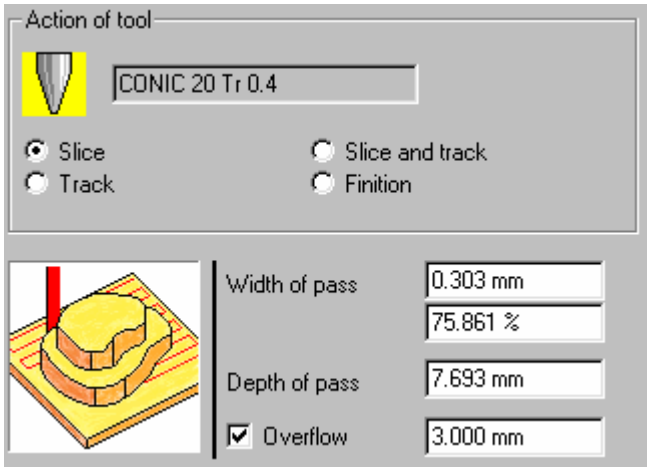
Use the icons on the right to add, delete or modify the tools included in the sequence.

Overthickness: leftover material allowed for the finishing process (here a value of 0.2 mm)

Approach: specific value for the Z reference, when the tool is above the material. If the engraving is very low, it allows the tool to move to this Z in fast speed.

Specific tab for each step

For each step, you define in the corresponding tab the parameters that determine the **action of the tool**, with four possible options:



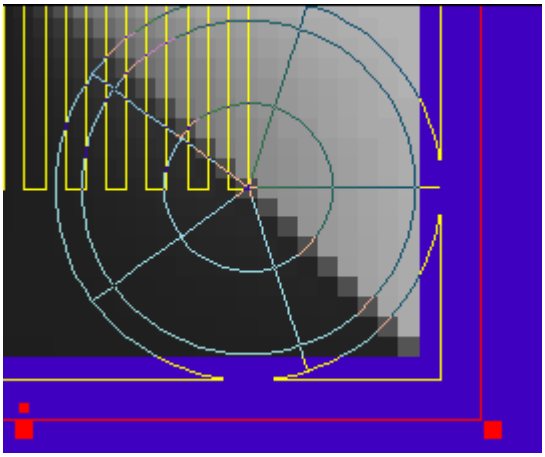
Slice

The object is cut into slices at the defined **Width of pass** (the sweeping is constant in Z and stops at the limit of the object).

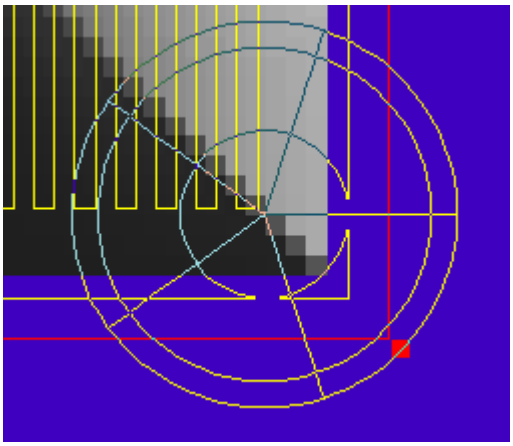
Track
The tool follows only the profile of the object.
Slice and track
After having machined a slice, the tool follows the profile of the object until it reaches the next slice.
Finishing
The tool goes everywhere in a continuously.

Width of pass: distance between passes (in mm or in %).
Overflow determines the global machining mask for the TypeArt object. Without overflow, the edge of the tool won't go beyond the edge of the TypeArt. With some overflow (less or equal to the radius of the tool), the toolpath will go beyond.

Warning: the overflow and its value are related to the action required by the previous pass. The overflow cannot be larger each step, for example (the OK button remains grayed until you correct the values).



Machining without overflow



Same machining with overflow equal to the radius

TOOL PARAMETERS

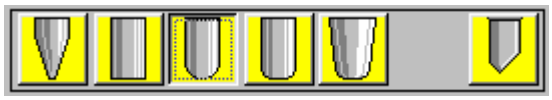
Tool definition

These parameters are displayed and/or entered when a tool is created/modified in the database.

WARNING
Some parameters are shown in the dialog box for all tools, even if they do not apply to the current tool.

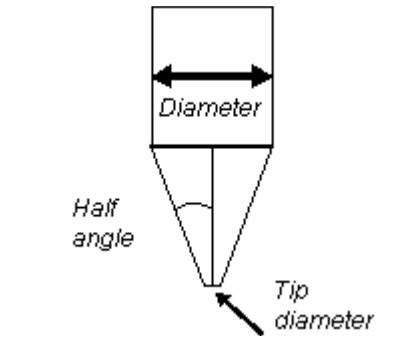
- **Tool's shape** tab

Supplier: name of the tool supplier.
Reference: tool reference of the supplier.
Family of tools



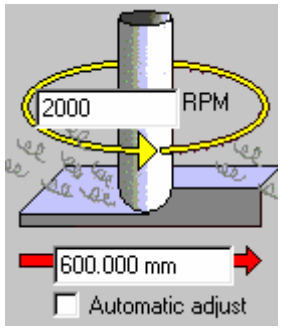
The shape of the tool you are defining must be selected among preset families (cones, cylinders, balls, etc.). The shape parameters are determined by this choice.

Example of a conical tool:



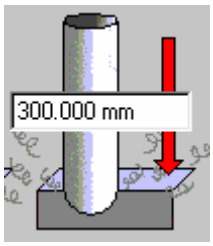
Diameter: diameter of the tool used.
Half angle: this parameter corresponds to half of the value of the angle.
Tip diameter: value of the diameter at the tool tip

Feed speed and rotation speed of the tool



The rotation speed is given in number of rotations per minute. The nominal feed speed is indicated in mm/min. If the **Automatic box** is ticked, plunge and descending feed rates are automatically adjusted in function of the values.

Plunge feed rate

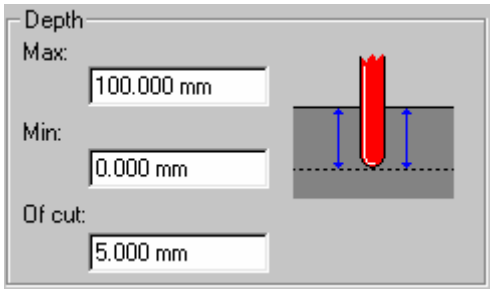


Vertical job speed in material.

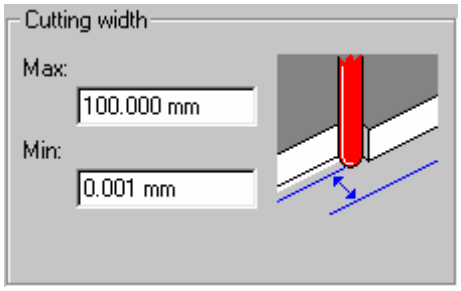
Tool index

See **Turret** tab.

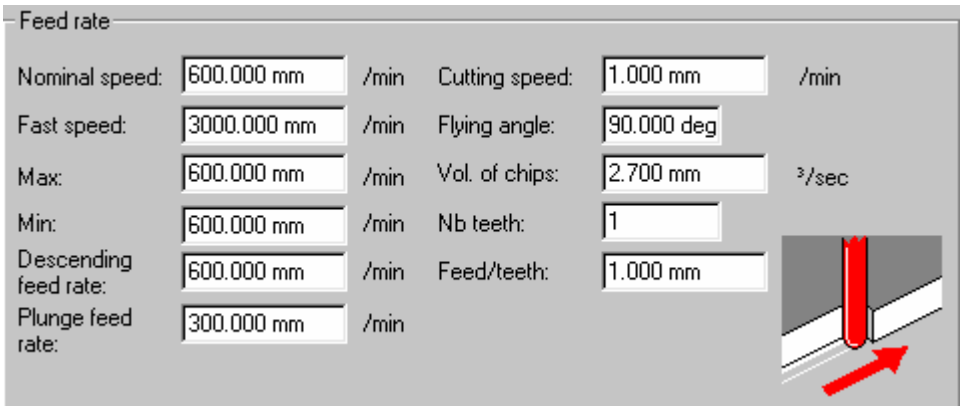
- **Technologies** tab



The depth cutting is given by default and cannot be exceeded when the toolpath is defined.



These values indicate the **distance between passes** specified in toolpaths.

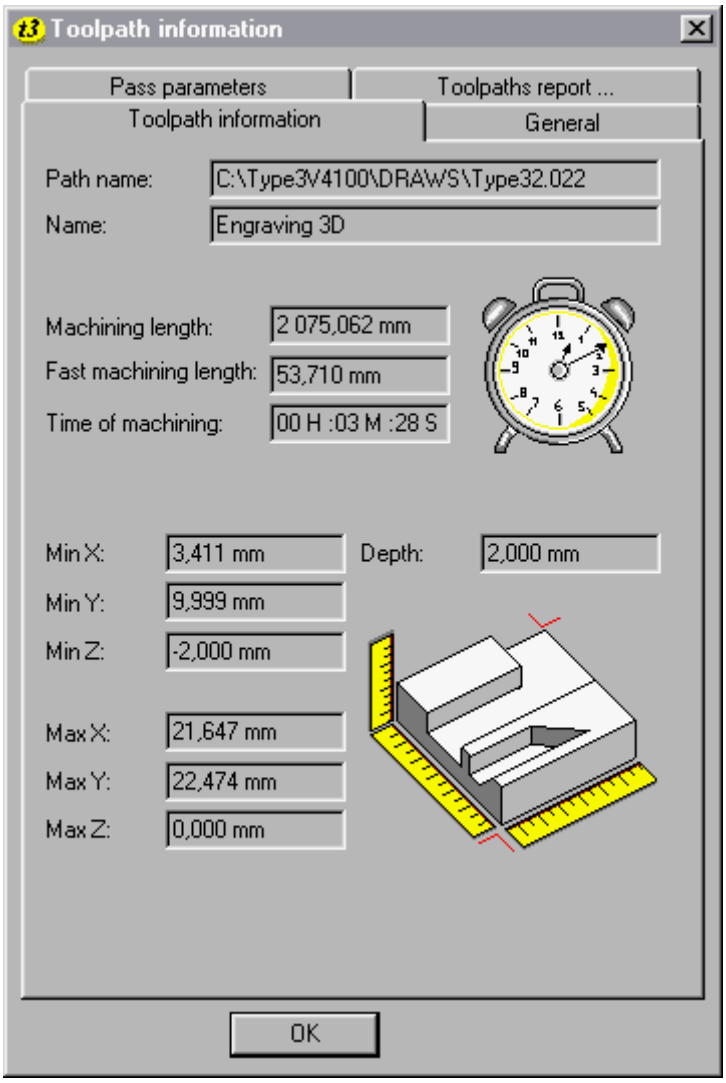


The fast speed is used for the lift-off of the tool out of the material. The nominal and fast speeds are useful for the calculation of the machining duration.
The **plunge feed rate** is the vertical job speed, the **descending feed rate** is the speed of 3D machining.
In some cases, the right side gives the possibility to enter technical parameters of the tool in order to compute the speeds.
The **flying angle** is used for the cutting. The tool will descend with an angle and will apply a lower stress on the material.

TOOLPATH INFORMATION

Display

You can see the file of a tool path at any time. Select therefore the desired toolpath in the dialog box **Toolpath list**. Click on the mouse right button to open the context menu, select **Information…** and click the mouse left button. The dialog box **Toolpath information** is displayed on the screen.



Select the tool path desired to see the following information:

Path name
Directory and file name of the tool path.
This file has the same name as the document, with the extension xxx (XXX = numbers corresponding to the order of creation of the path).

Name
Machining mode (plotting, 2D engraving, etc.)

Machining distance
Total distance machined by the tool.

Distance of fast displacements
Distance of fast displacements executed by the tool above the material (ascents, descents, etc.).

Machining time
Total time of machining in accordance with information given in the tool data base **+** command.

Min X MinY Min Z
Minimum coordinates of the tool path.

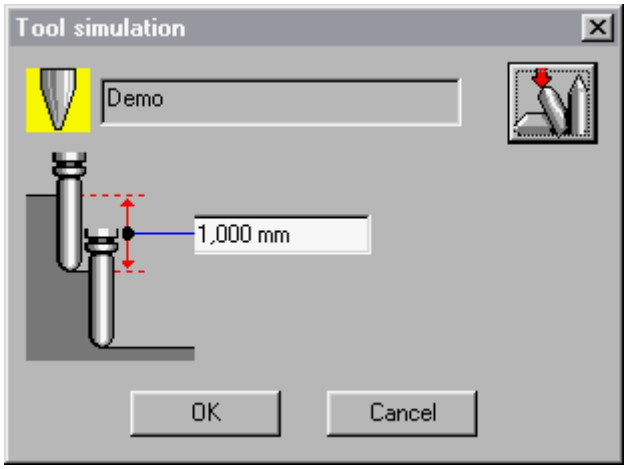
Max X MaxY Max Z
Position of last point

Depth
Total depth of engraving defined during the tool path computation.

Displaying the tool cursor

More than a measuring tool, this function permits you to check if the tool, that you will use for the machining fits the created shape.

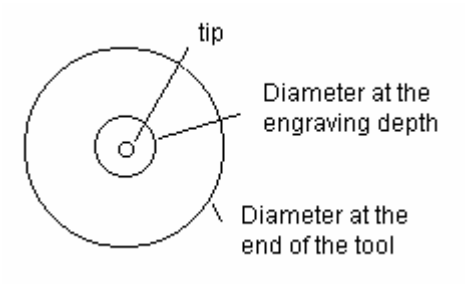
In the **Measuring toolbar**, click on the icon **Tool simulation** . The corresponding dialog box is displayed on the screen.



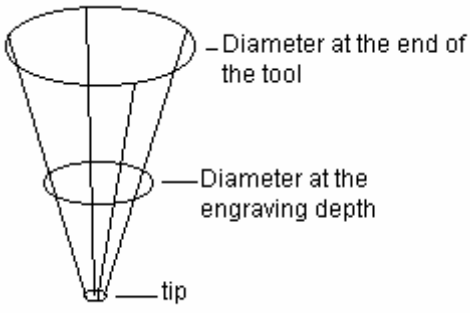
Choose the tool in the dialog box. Click therefore on the **Tools DataBase** icon . Enter the depth of machining in the field of the dialog box.

The cursor representing the tool is characterized by three circles:

- **For conical tools**



2D View

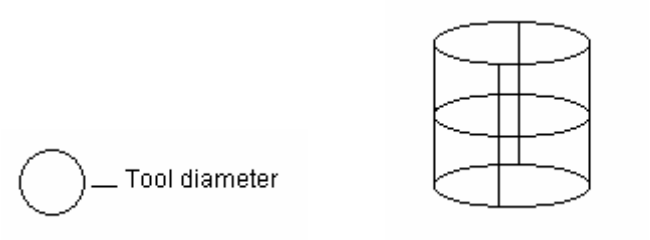


3D Iso View

Remark

Only the diameter at the engraving depth is variable.

- **For cylindrical tools**

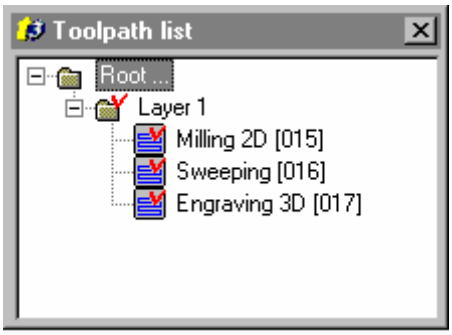


2D View *3D Iso View*

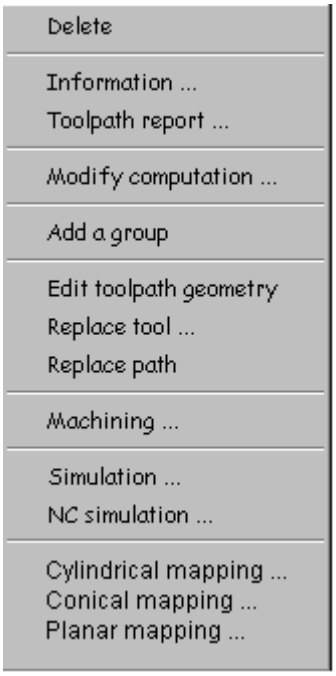
Move the cursor which represent the tool in all directions; to see for instance if the tool will machine the whole piece.

Simulation display

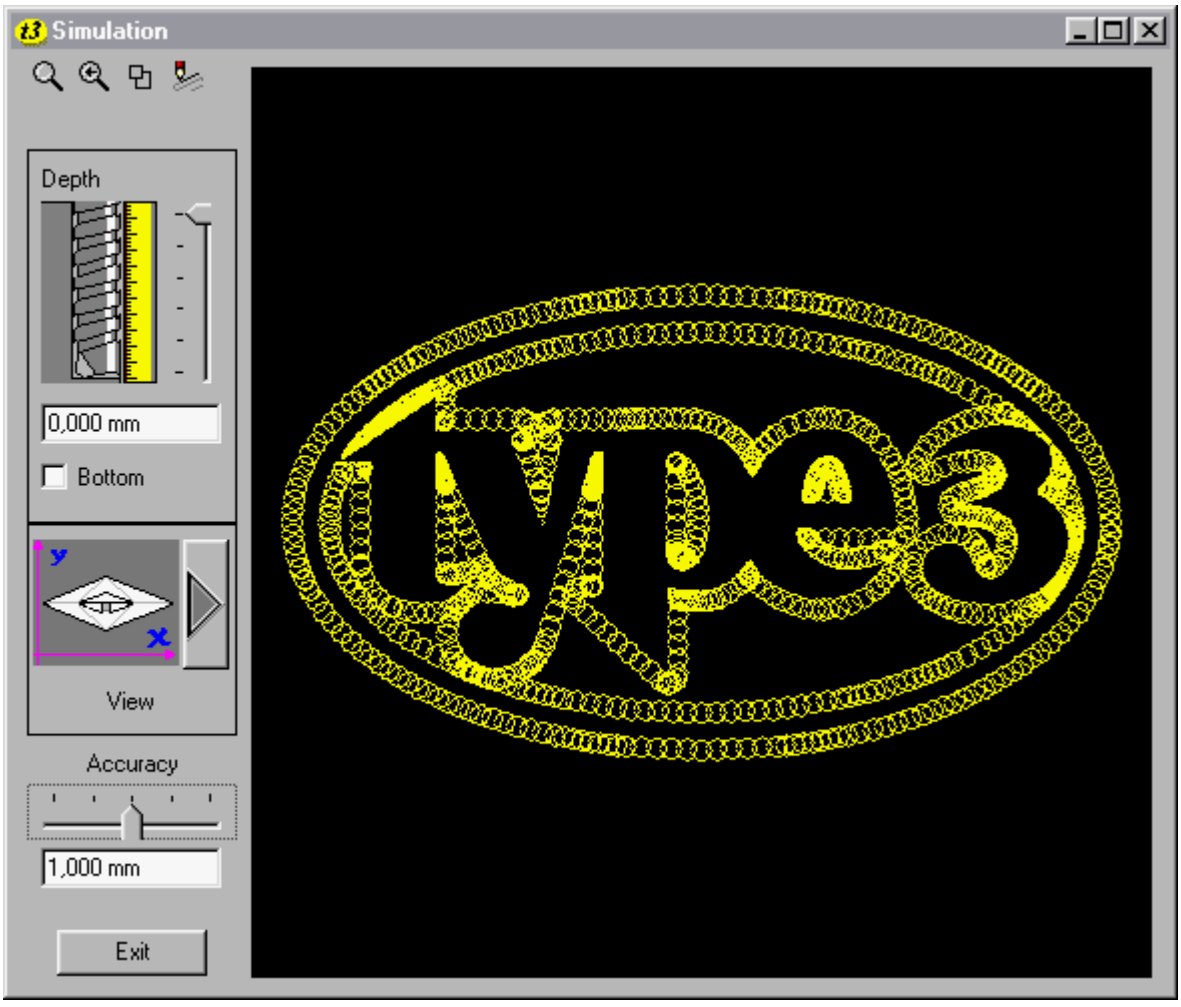
The display of the created toolpath is a nice feature of the CAM module. This function is very useful for showing a simulation of tool displacements, especially for conical tools. You will have a representation of the manufacturing process, in other words, the material removed by the tool. In the toolbar of the CAM module, click on the icon **Toolpath list** . The corresponding dialog box is displayed on the screen.



Select the toolpath to be simulated, and click on the mouse right button to open the context menu.

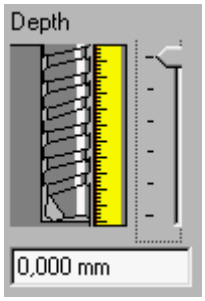


In the context menu, click on **Simulation…**. The corresponding dialog box is displayed on the screen.



By default, the dialog box **Simulation** displays the toolpath at the top of the cutting (in the example shown above, the height is Z=0).

Nevertheless, you can modify the visualization height by using the view depth cursor or by entering the desired value in the numerical field located below.

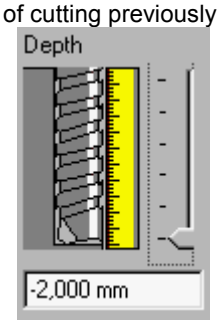


Remark

You can set the viewing depth using the cursor: the numerical field value will be automatically updated and the view will be refreshed.
In case you want to set the depth numerically, double-click in the field, enter the desired value and validate by clicking in the other numerical field of the dialog box Simulation**. The toolpath visualization will be automatically refreshed.**

View from the bottom

To see what happens at the bottom of the engraving (or on the other side of the object when using the cutting function), use the depth cursor or double-click in its numerical field, enter a negative value which is **– 2 mm** in our example (and which corresponds to the depth of cutting previously set) and click elsewhere to validate.

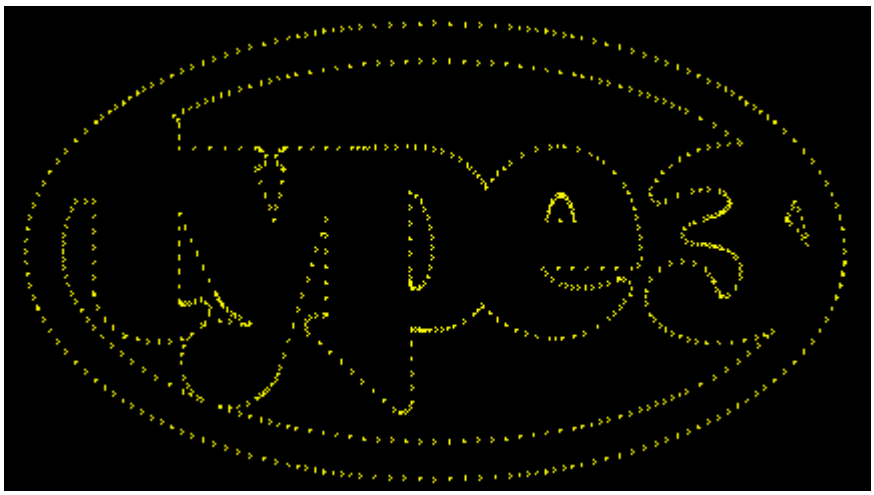


This option is especially useful to check the machining at the bottom of the engraving.

Remark

The simulation appears thinner at 2 mm depth than at 0 mm because the tool used is a conical one.

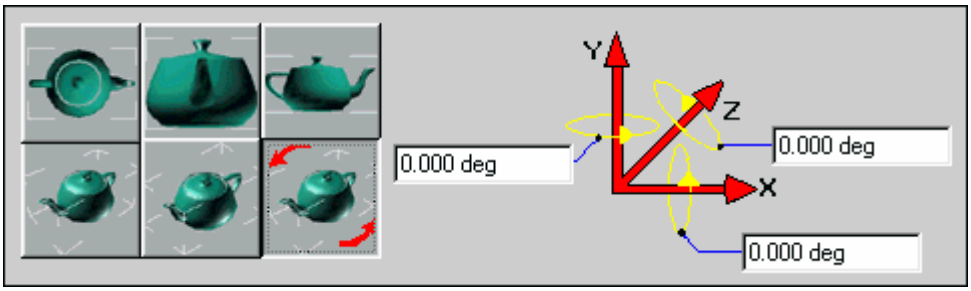
Simulation at Z=-2mm



Surface density
To increase or to reduce the display density of the tool cursor, move the cursor **Accuracy** of the dialog box **Simulation**, or double-click in the numerical field below, enter a value and click in the other numerical field to validate it.

View angle of simulation

To modify the view angle of the toolpath simulation, click on the icon of the dialog box **Simulation**. The toolbar of the view options is displayed on the screen.



Double-click on the desired option. The visualization is automatically refreshed.

Remark

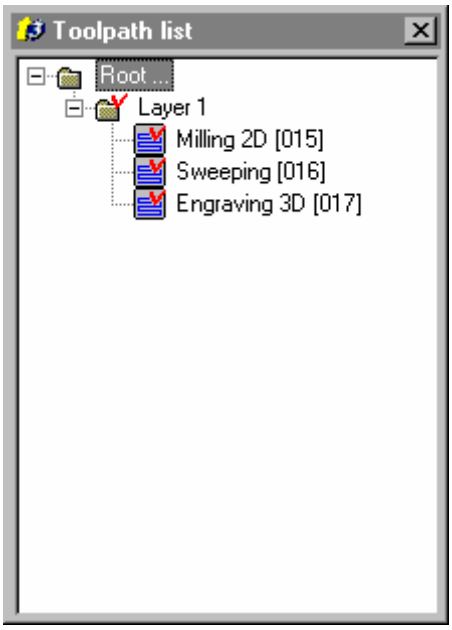
If you choose like in the example shown above the Free 3D view **(symbolized by the** 3D **icon), you must enter the values that define the view angle in the corresponding numerical fields.**

NC SIMULATION

Introduction to the NC Simulation

It is possible to display the toolpath accurately on the screen. You can therefore have a good representation of the machined object.

In the toolbar of the CAM module, click on the icon **Toolpath list** . The corresponding dialog box is displayed on the screen.



Select the toolpath to be simulated, and in the context menu, click on **NC Simulation…**. The **Simulation** box appears on the screen

The rendering happens the first time you enter this dialog box, and each time you modify an option.

NC Simulation tools

Push this button to launch the physical simulation **with** the visualization of the tool and the toolpath.

Push this button to launch the physical simulation **without** visualizing neither the tool, nor the toolpath.

Push this button to launch the physical simulation **with** the visualization of the toolpath and **without** the visualization of the tool.

Undo.

Pause. Stops the simulation and displays the part already calculated. If you ask for a new simulation, it will start from this breaking point.

Precision level of the NC Simulation (**1** for the best result, **3** for a quicker one).

Realistic rendering of the part in **grayscale** or in **2 colors** (gray and yellow – metal effect**).**

Back to the original material.

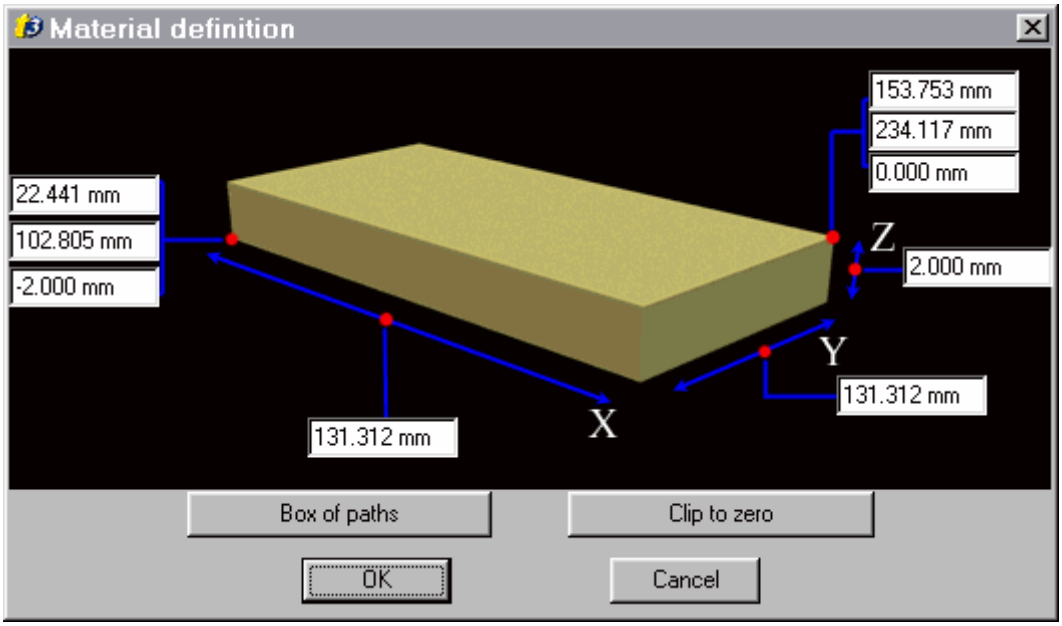
Zoom.

Push this button if you want to display the box **Definition of material** in order to define the parameter again.

Push this button if you want to transform your object into a **TypeArt** object.

By clicking on the left mouse button, you can rotate the 3D view. The right mouse button allows you to zoom the object in or out.

Material definition
To define the material before launching the simulation, push the Ctrl key when you click on **NC Simulation**. The corresponding dialog box is displayed on the screen.



- Click on **Box of paths** if you want to machine the whole object.
- Click on **Clip to zero** if you want to put the origin of the Z axis to zero (Z = 0 corresponds to the upper part of the material).

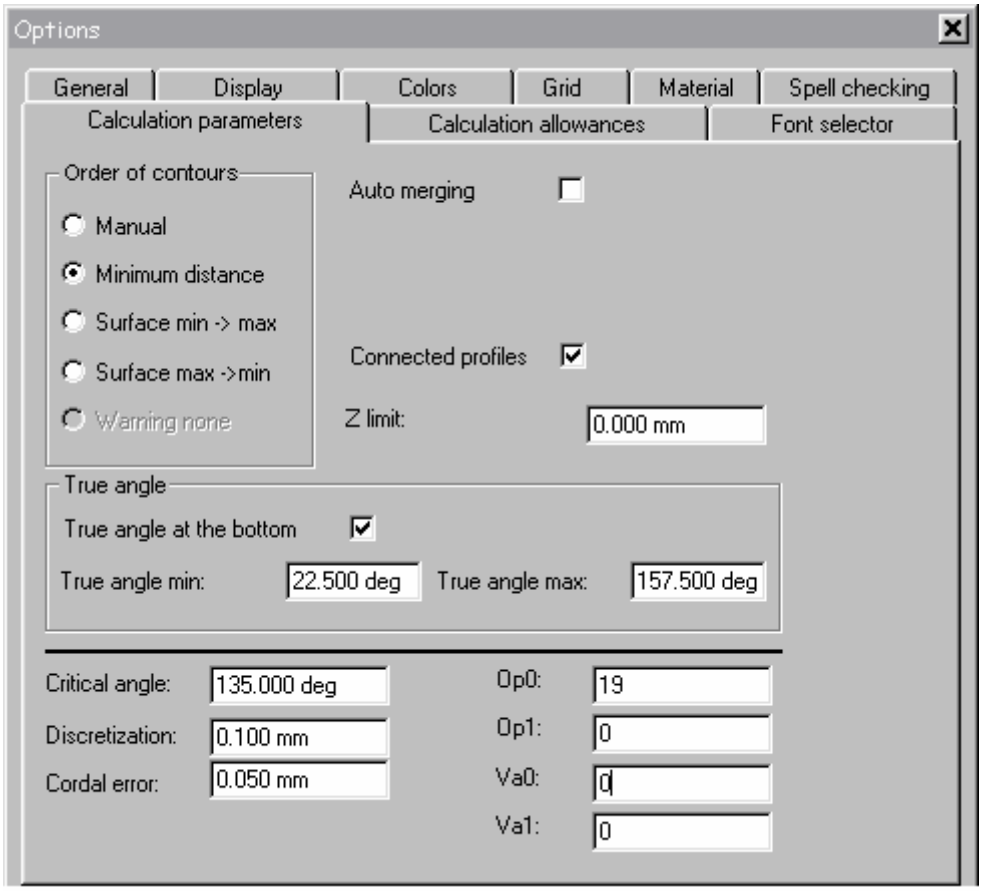
Then, click on **OK**. The **Simulation** box is displayed on the screen.

CALCULATION PARAMETERS OF THE TOOLPATHS

It is generally needed to restart the software so that the post-processor applies the modifications made in the Calculation parameters.

Calculation parameters

You can reach these parameters by hitting the **F10** key in the CAD module. In the **Options** dialog box, select the tab **Calculation parameters**.



Machining order (Order of contours section)

The software has been configured by default to reduce the time of machining. For certain jobs, like cutting, you have at your disposal a second level of sorting where you can modify the order of contour management.

Manual
When sorting in the manual mode, the starting point and the order of contour selection (creation) defined in the CAD module will remain the same.

Minimum distance
Default mode: the machining of contours is done by the order of proximity (X min to X max) in order to minimize tool path time.

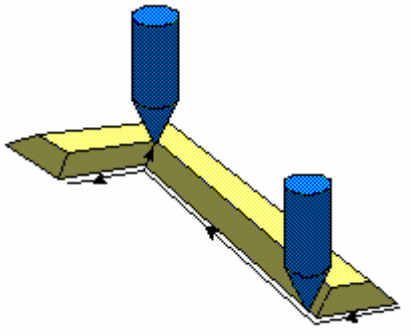
Surface min → max
In this mode, machining of the small surfaces is done first (inside the letter, for instance).
This mode is automatically selected when you choose 2D or 3D cutting.

Surface max → min
The large surfaces are machined first.

Warning none
Function not implemented.

True angle at the bottom

Select the option **True angle at the bottom** to generate a sharp angle at the bottom, in 3D machining (for the angles included in the range)



This option is useful in manufacturing dies, because the angle at the bottom of the die produces the angle at the top of the final part.

True angle min
Minimum value above which a true angle is made.

True angle max
Maximum value under which a true angle is made.

Multipass (Op0)

Op0 = 3 Multipass or multi-step (in Z) is not active.

Op0 = 17 Multipass is active. For the “Sweeping” toolpath, a rotation of 60° is done automatically between each pass.

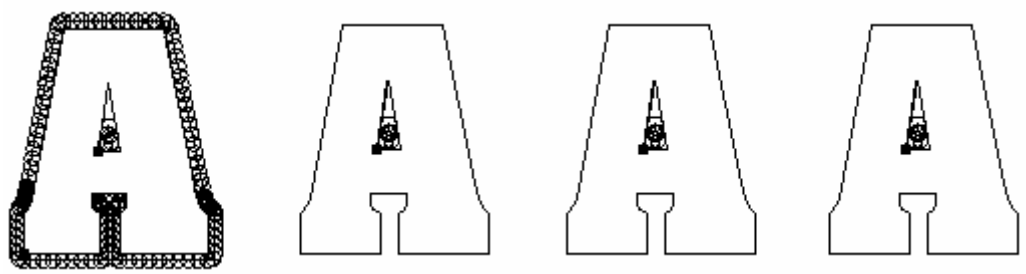
Op0 = 19 Multipass is active. For the “Sweeping” toolpath, a rotation of the value of the parameter n°19 (Options dialog, Calculation allowances tab) is done automatically between each pass.

By default, multipass is active, without any rotation between each sweeping step (Op0 =19 ; parameter n°19 = 0).

Cutting mode for contours (Op1)

Standard

Op1 = 0
The smallest surfaces are machined first.



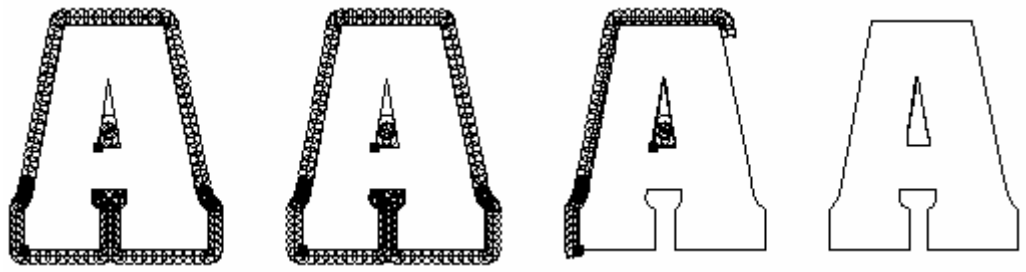
If the proposed default mode sorting of contours is not sufficient for optimizing your tool paths, you have at your disposal supplementary options, which improve the time of machining.

Enter in the field **Op1** the number 1 or 2 according to the type of cutting you wish to do (see below).

Cutting letters

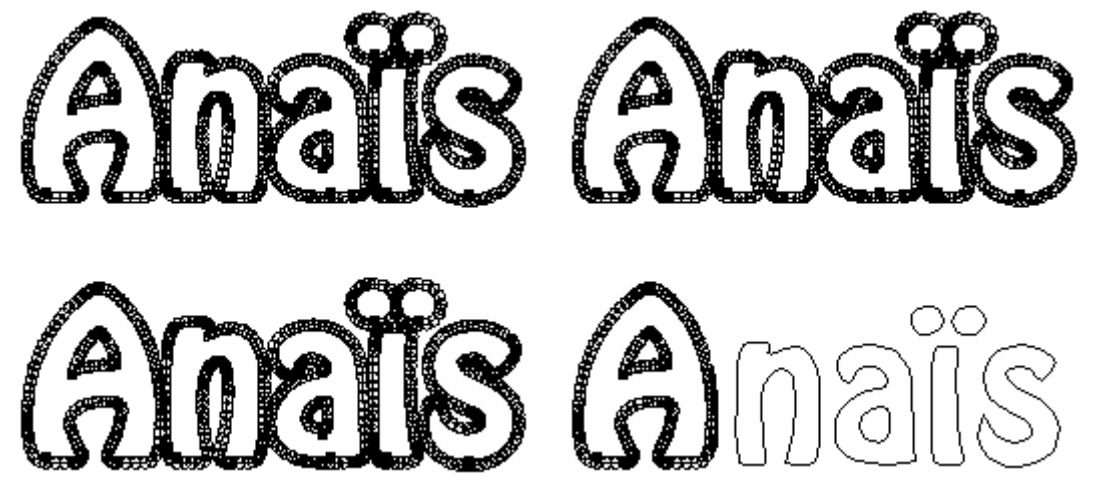
Opt1 = 1
All the objects are independent and will be cut one after another (in each case the internal contour will be cut prior to the external one). The calculation of trajectories is faster since the software does not execute the function concerned with the collision management of tools.

It is, therefore, necessary before beginning to leave enough space between the objects to avoid any possible collisions.



Cutting words

Op1 = 2
The order of selection in the CAD module remains the same. The cutting operation is made piece by piece (in each case the internal contour will be cut prior to the external one) with the collision management activated.



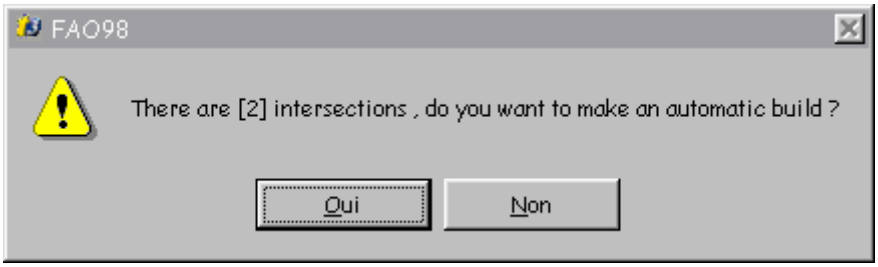
This option is recommended to cut pieces that do not require a complete cut between objects.

Op1 = 3
The initial order given to objects (during creation or selection) is kept.

The cutting is done part after part, optimizing the order internal/external.

Other parameters

Auto-merge
If checked, in the case of split contours including intra- or intersections, the machining module tries to find a global relevance for all contours and reconnect them. You're asked a confirmation.



The proper use of the auto-merging depends of the splitting and complexity of contours.

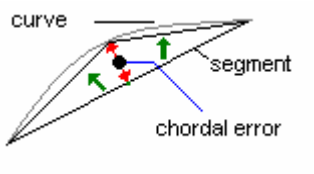
Connected profile
Machining continuously without lifting the tool when changing contours (optimizes time of machining when contouring)

Z limit
Z limit for 3D movements outside of the material (in particular true angles). In the case of tools with flat end, it avoids to see where the tool stopped its work. The resulting object will be smoother.

Critical angle
The value beyond which a sharp angle will not be done.

Discretization
Maximum length of the segments generated when the **Output curves as segments** box is checked in the window called by the **Parameters…** button of the **Machining** window.

Chordal error
Maximum height of the arcs generated when the **Output curves as segments** box is checked in the window called by the **Parameters…** button of the **Machining** window.

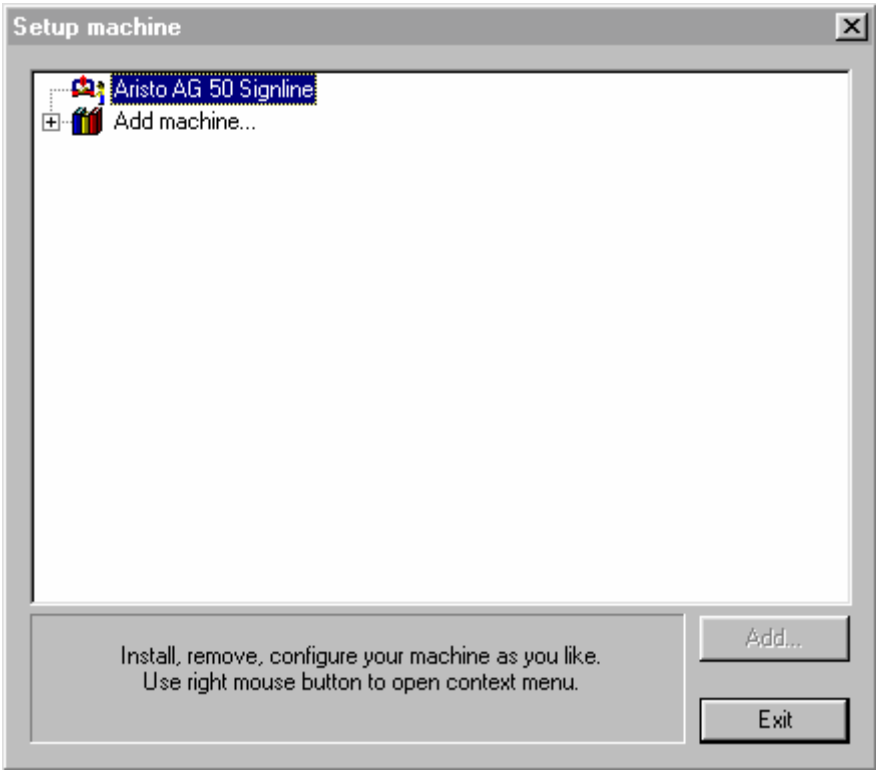


Va0, Va1
Reserved parameters, used for optimization algorithms.

MACHINE SETUP

Installation of the machine

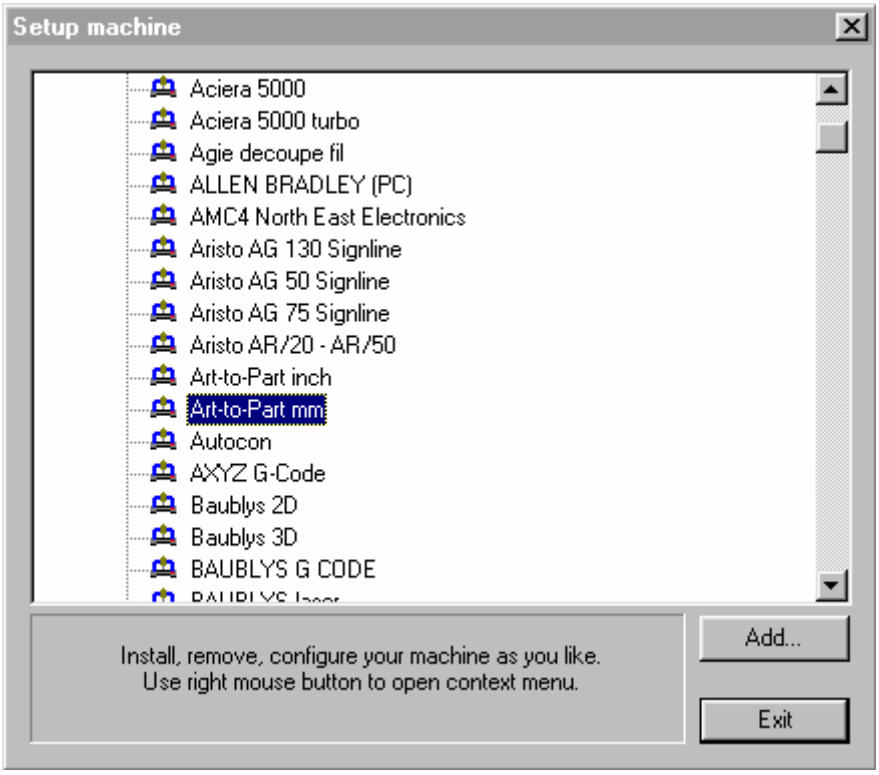
To set up your software for your engraving machine, select **Setup machines…** in the **Machining** menu. You can do this in the CAD module or in the CAM module**.**
The dialog box **Setup machine** is displayed on the screen.



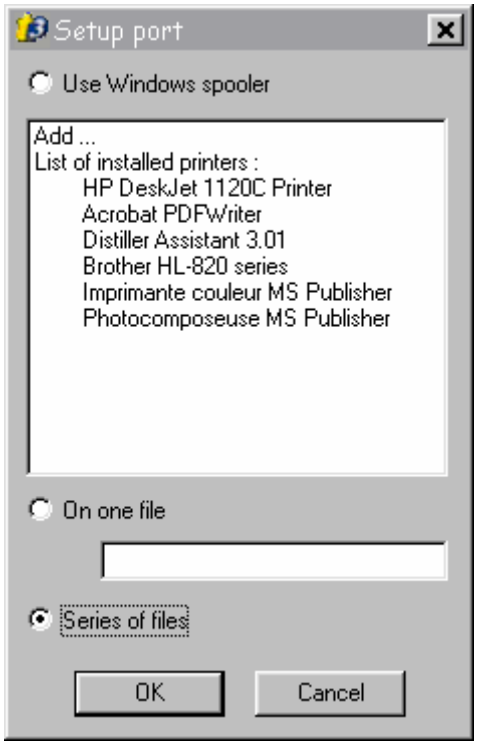
Double-click on **Add machine…** or click on the icon ⊞ to see the list of available machines. Select the kind of machine you want to install.

- **HPGL machines**: Cutting plotters, laser, etc.
- **ISO / CNC machines**: Machining centers, etc.
- **Engraving machines**: Roland, Mimaki, Mecanumeric, etc.
- **All machines types**: Complete list.

Select then the machine.



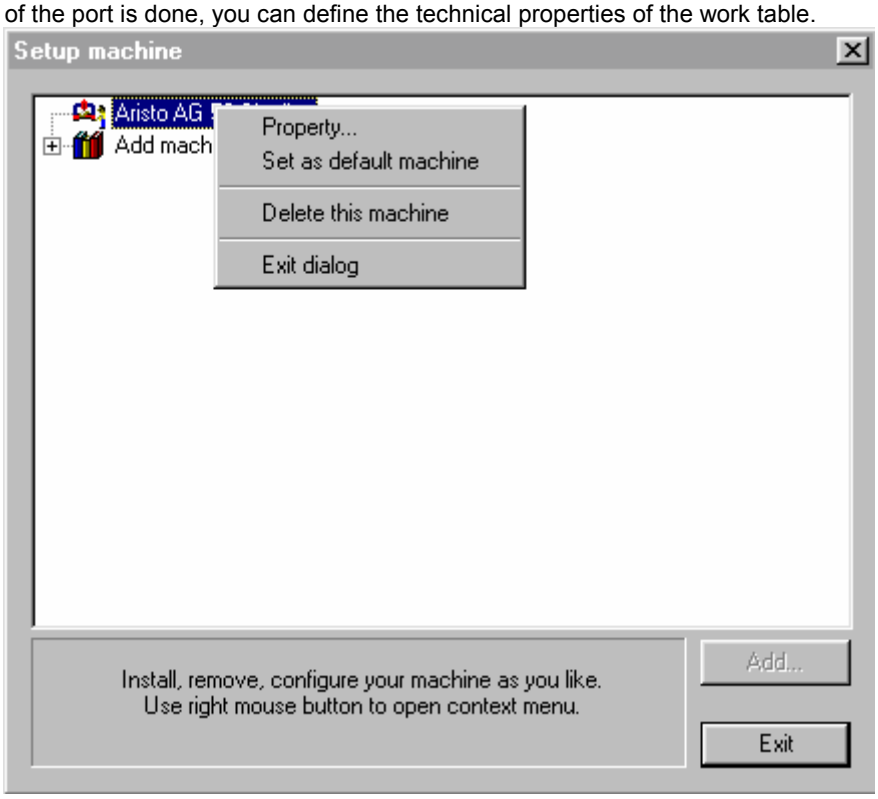
Click on the button **Add...** The dialog box **Setup port** is displayed.



The setup of the port is described in the section **Sending to the output machine**.

Sending to the work table

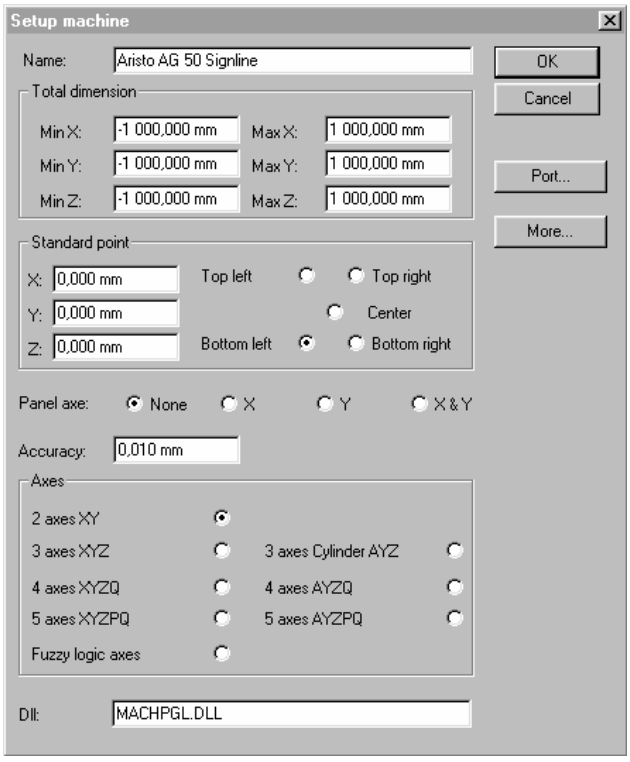
Once the setup of the port is done, you can define the technical properties of the work table.



Remark

You can configure your machine at any time by clicking with the mouse right button on the desired machine. Select the option Properties… **in the context menu that is displayed on the screen. The dialog box that are explained in this chapter will appear.**

The second dialog box **Setup machine** is displayed.



You have to fill in all information relative to your machine.

Name
It is the name of the machine that appears in the list of the installed machines.

Total dimension
It is the dimension of cutting area. A warning signal will appear each time the machining goes outside this defined zone.

Standard point
Special assembly point (if any) on the machine (vice, ...)
Choose and click on the choice that corresponds to the desired position.

Panel axis
Not yet available

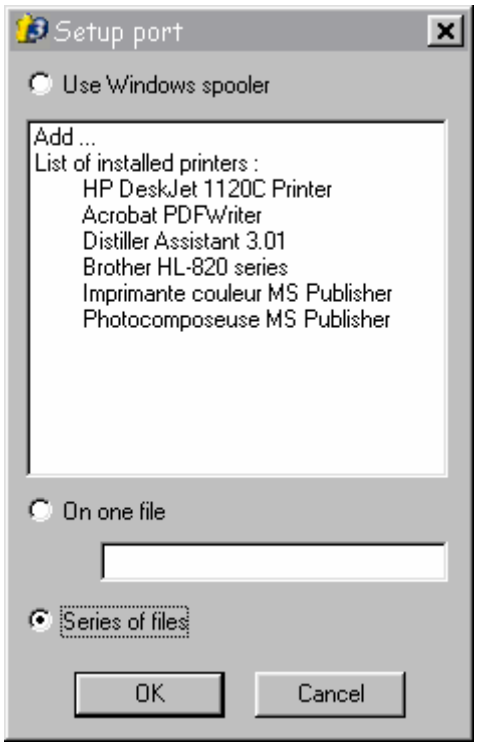
Axes
Check the option that corresponds to the capabilities of your machine tool. The options with 4 and 5 axes are not yet available.

Accuracy
Accuracy of calculation used by default during the toolpath calculation.

Dll
It is the reference library that is used by the post-processor to generate the code of the machine.
Don't modify it!!!

Sending to the output machine

The setup of the output machine has to be done after the setup of the machine.
In the dialog box **Setup machine**, click on the button **Port…**.
Select the desired options in the dialog box **Setup port** that is displayed on the screen.



Series of files
This option is recommended if no machine is connected. It allows you to define the name of the machining files that will be generated automatically based on a prefix and that you will be able to edit.

Use Windows spooler
If your machine is directly connected to your computer, you can use the Windows spooler to directly send the tool path to the machine without loosing time (see Windows documentation to configure the spooler).

On a file
To write a tool path on one and only file (on your computer or on a network device). Define its name. You have only one possibility and you can't modify it.

The dialog box **Setup port** allows the installation of a printer without having to go through **Windows**.
Double-click on **Add...** in the dialog box **Setup port**.

Just read the Windows User's Guide to install a printer respecting the kind of printer according to:

- the manufacturer
- the printer.

If your machine is connected to the parallel port, you don't have to configure the port. On the contrary, if it's connected to the serial port, you have to configure the port according to the specifications given by the manufacturer of your machine.

Example

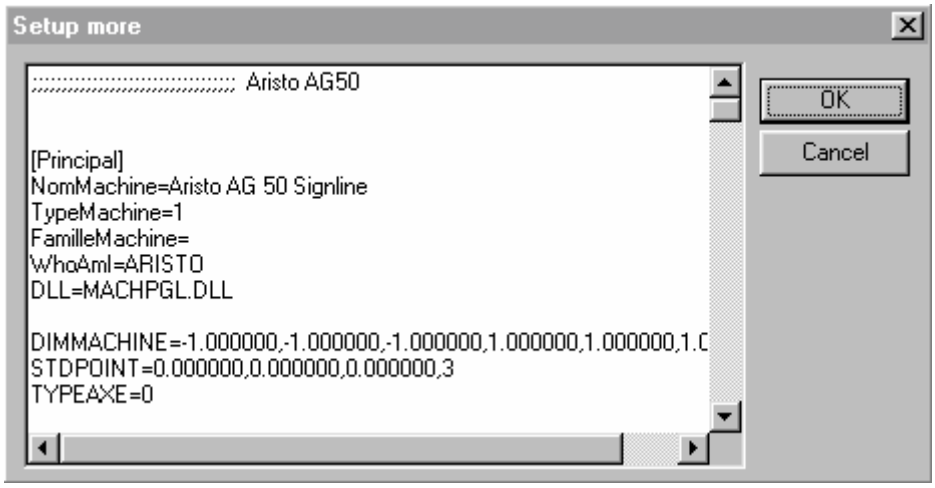
Port parameters

- Bits / sec.: 9600
- Data-bits: 8
- Parity: none
- Stop-bits: 1
- Input / output control: Xon/Xoff

Post processor characteristics

Modifying the characteristics of the post processor is made from the dialog box **Setup machine**. Select with the mouse right button the machine to be configured and choose the option **Properties…** in the context menu displayed on the screen.

Click on **More...**
The dialog box **Setup more** is displayed on the screen.
You can now modify the characteristics of the processor within this box.



Warning!!!

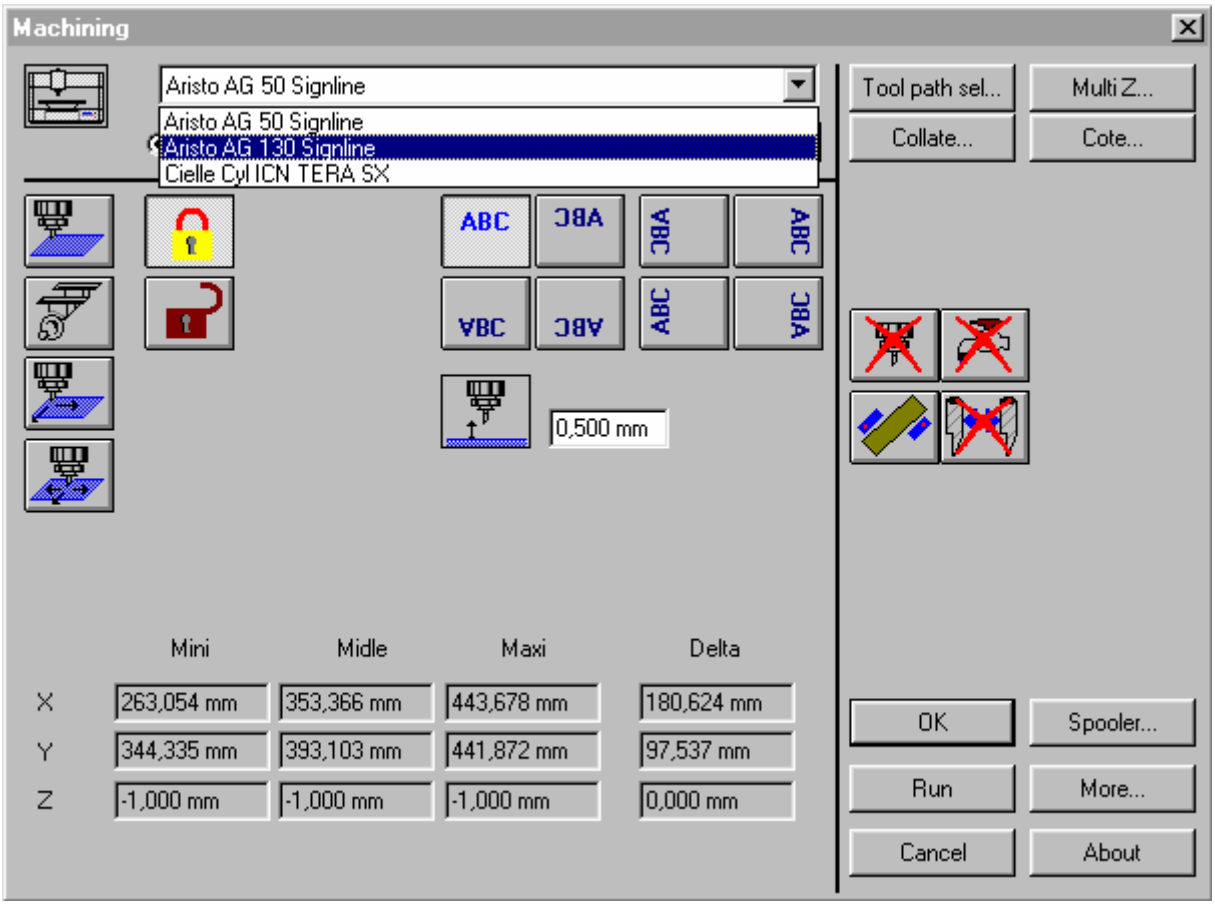
These parameters are optimized, before doing any modifications of the characteristics of your machine post processor, we highly advise you to contact your reseller.

Starting of machining

When the machine is ready, you can prepare the machining program.

Select the toolpath in the dialog box **Toolpath list**, click the mouse right button and select **Machining…** from the context menu that gets displayed on the screen. In both cases, the dialog box **Machining** is displayed on the screen.

Choose the machine in the list and enter the parameters according to the type of machining to be done and according to the material at your disposal.



The **Spooler…** button allows to display the Windows Spooler queue.

Be careful before clicking on this button: it is of the utmost importance to check the Windows Spooler **option in the dialog box** Setup port**.**

The **More…** button opens the dialog box **Other parameters** which concerns the characteristics of the post-processor. You can find the same button in the dialog box **Setup machine**.

Choice of the machining file output

Test
To display only a part of the file on the screen. Nothing is sent.
Port
To send directly on the connected machine (see **Setup machine** for configuration).
File
This command creates the machining file that you will be able to use according to the installation of your machine. To make the search easier, the name and the path of the file are written on the front.
Change... Use this command button to modify the name, path and number of the file (U00 to U01 for instance).

Positioning the piece

- **"Maximum X, Y and Z" "Middle X, Y and Z" "Minimum X, Y and Z"**

The above coordinates are those of the tool path: the lowest point of engraving, the center point and the highest point.

- **"Delta X,Y and Z"**

These coordinates correspond to the difference between the Max and Min coordinates. Actually, they correspond to the real size of the engraving.

- **Locking and unlocking of the coordinates**

Locked
If this function is activated, **X**, **Y** and **Z** coordinates are determined by the reference point defined in the CAD module. In this case you cannot change these values.

Unlocked
If this function is activated, you can modify the **X**, **Y** and **Z** coordinates (minimum, maximum and middle) in order to reposition your engraving. If you modify one of these values, the others will be automatically recalculated.

Four predefined piece positioning models are available.

REMARK
In the four next examples, the technical word Plaque **refers to the working area in the CAD module.**

Plaque-origin at the upper left corner
If you click on this icon, the plaque-origin will be positioned at the upper left corner of the engraving area.

Plaque-center at the center
If you click on this icon, the plaque-center will be positioned at the center of the engraving area.

Free plaque-origin
If you click on this icon, you can position the upper left corner of your plaque on any defined point of the engraving area. You just need to enter the desired values in the fields displayed at the right side of the icon.

Free plaque-center
If you click on the icon, you can position the center of your plaque on any defined point of the engraving area. You just need to enter the desired values in the fields displayed at the right side of the icon.

Remark

The 2 latter options allow to position the active and the machine surfaces and as a consequence to avoid eventual possibility of overflow when machining. These problems can be solved by 3 ways:

- **Redefining the position of the piece in the Machining dialog box,**
- **Changing the dimensions of the engraving surface (Properties option),**
- **Redefining the dimensions of the active surface in the CAD Module (Modify menu → Selection).**

Positioning the toolpath

The **Machining** dialog box allows you to reposition the tool paths without having to recalculate them.

is the default option.

Activate this icon to position the tool path like it has originally been conceived.

Activate this icon to make a centered rotation of the tool path in relation to the plaque.

Activate this icon to make a centered rotation of the tool path in relation to the plaque and then a 90° anti-clockwise rotation.

Activate this icon to make a 90° clockwise rotation of the tool path.

Activate this icon to make a centered rotation of the tool path in relation to the plaque.

Activate this icon to make a 180° rotation of the tool path.

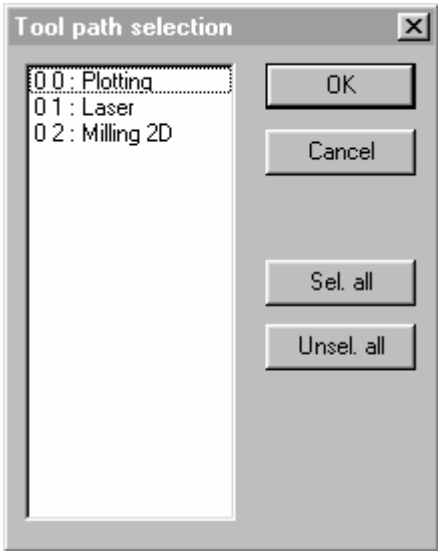
Activate this icon to make a 90° anti-clockwise rotation of the tool path.

Activate this icon to make a centered rotation of the tool path in relation to the plaque and then a 90° clockwise rotation.

Z between contours
If you click on this icon, you can directly access to the height (Z value) of the tool between contours. This can also be done within the **Machining parameters** dialog box (therefore, see the **Parametrizing positions and machining limits** section in the same chapter).

Selecting the toolpaths to send

Click on the button **Tool path Sel…** in the **Machining** dialog box to choose in the list the tool paths to be machined.



Collate

You have 3 possibilities to group - or collate - the tool paths. Therefore, click on **Collate…** in the dialog box **Machining**.

None
If you don't want to collate, you'll get as many machining files as toolpaths.

By layer
All tool paths of a same layer are sent in a same file.

All
All tool paths and all planes are regrouped and become one single operation on the machine.

Remark

Be aware that by choosing this option, the machine will stop after each machining so that you can change the tool.

Output curves as segments
To send only linear elements to the machine, circular elements are converted into small segments.

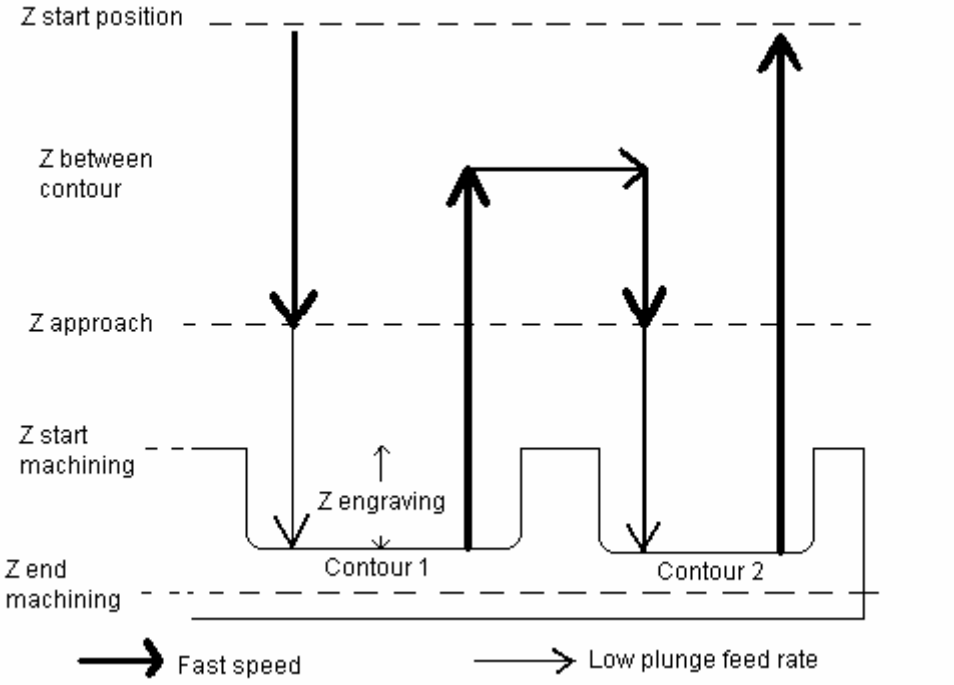
Initial position/Ending position
Position of the tool at the beginning and at the end of machining.

Change tool position
Coordinates where tool will be changed.

Z security
Set limit values to the tool in the Z direction respectively at the beginning and at the end of the machining.
For instance, if **Z security end of machining** equals –100, all tool movements under –100 won't be taken into account by the post-processor.

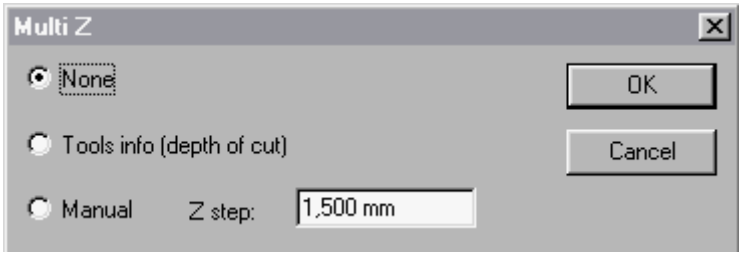
Z between contours
Position of the tool along Z axis between each contour (ascent of disengagement).

Z approach
Approach position along Z axis.
The value is relative to the surface to be machined.



Multi Z

This function is useful for machining the total depth in several passes. It is a complementary function of the information given in the tool path section (total depth in x times or depth of each pass).



None
Depth is done in one pass.

Tool info
Machining is done following the instructions given. In the **Tools DB** see **Depth of cutting**.

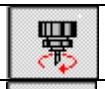
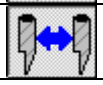
Manual
The machining is done with a cutting in Z following the step defined by yourself.

REMARK

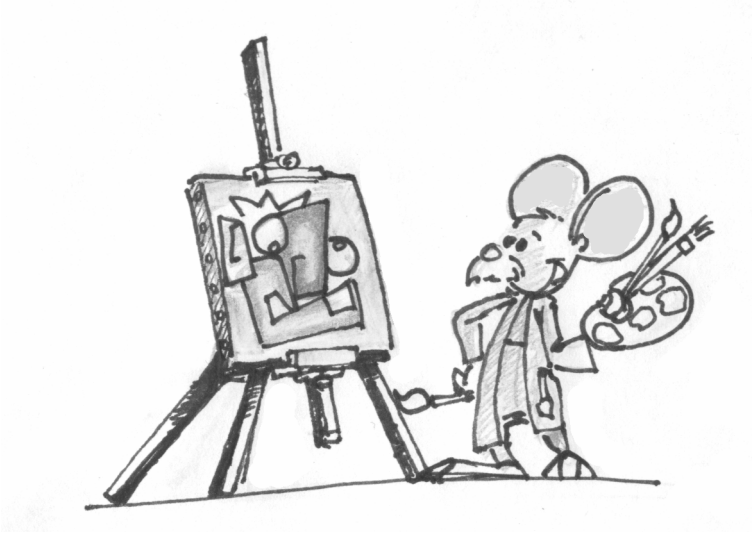
If you want the multi-pass option to be activated, you must be sure that the parameter Op0 is equal to 19. To access this parameter, choose Options… in the Edit menu (or strike the F10 key). The dialog box Options is displayed on the screen. Select the bookmark Calculation parameters to access Op0.

Machining window: option buttons

Four buttons are used to activate options sometimes available with some post-processors. Nevertheless these buttons have **no effect** with most post-processors.

Activated button	Deactivated button	Effect
		Activates or deactivates the pin rotation.
		Activates or deactivates the spraying (for cleaning or lubricating).
		Activates or deactivates the automatic plate loader.
		Activates or deactivates the tool loader (if any).

TYPEART MODULE

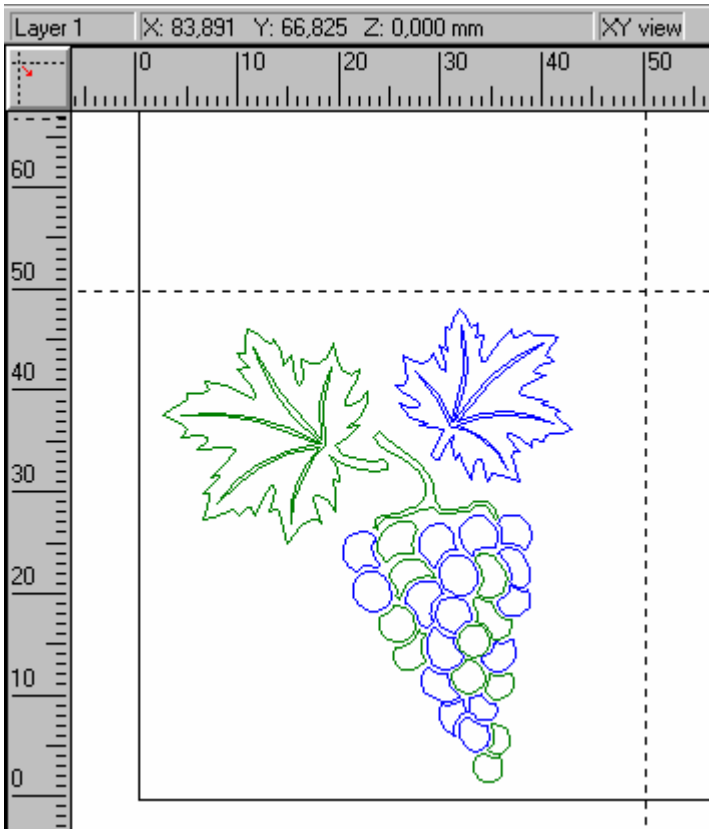


Drawing importation

Opening a new drawing

Place two guidelines, one vertical and one horizontal at 50 mm from the bottom left hand corner of the active area.

Retrieve a symbol from the symbol library, for instance **Grappe. syb**.



Decomposition of the drawing

Uncombine the drawing by clicking on the Uncombine icon , so you can work separately on each contour, and give to each a different shape.

Then only select the grapes by creating a selection window with the mouse and combine them.

Remark

You can perform all these operations (group/ ungroup, combine/ uncombine) under the TypeArt **module with the same icons as in the** CAD **module.**

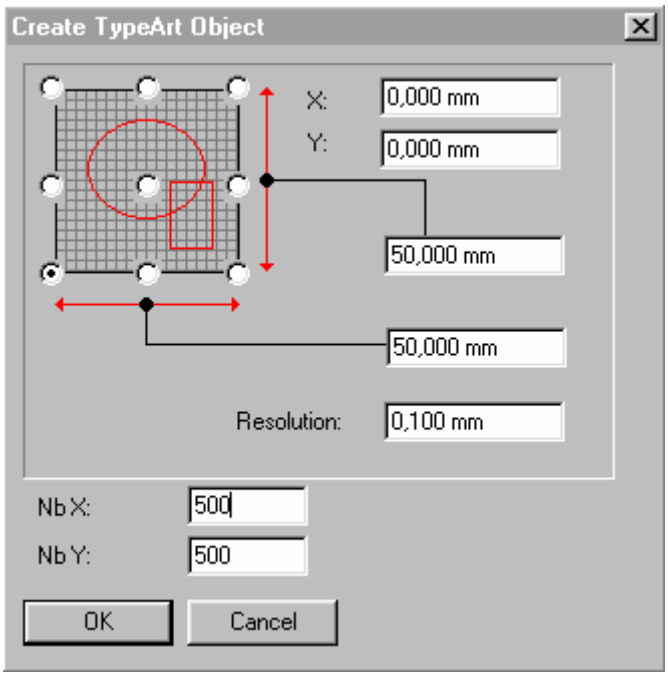
Saving your work

Before going to the **TypeArt** module, save your drawing under the name **grappe** and verify the directory (C:\Vision\draw\).

Running TypeArt module

Click on the **TypeArt** icon .

The window for creating a TypeArt object will be displayed on your screen.

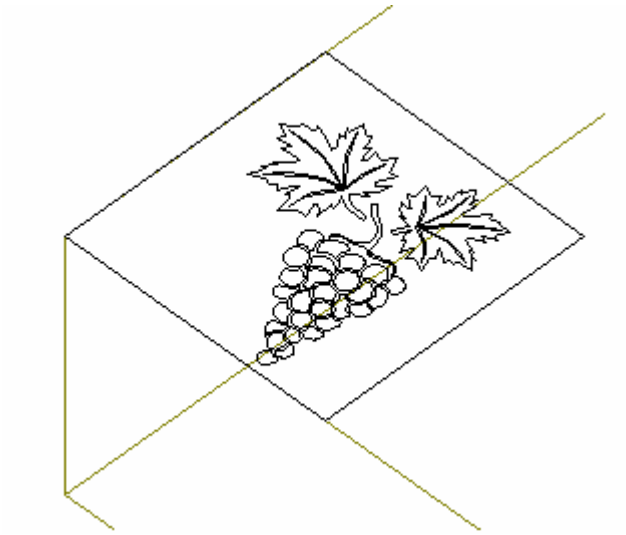


Choice of the parameters of the object

For the time being, we advise you to keep the default value at 200 for Nb X and Nb Y for a virtual test.

Later in order to get a better result in machining, choose a higher value (500 for a size of 50 x 50 mm, that will give you a resolution of 0.100 mm versus 0.25 mm for the default value).
In this example, maintain the default parameters and click on **OK**.

The **TypeArt** screen is displayed and the **3D ISO view** is automatically selected.



Remark

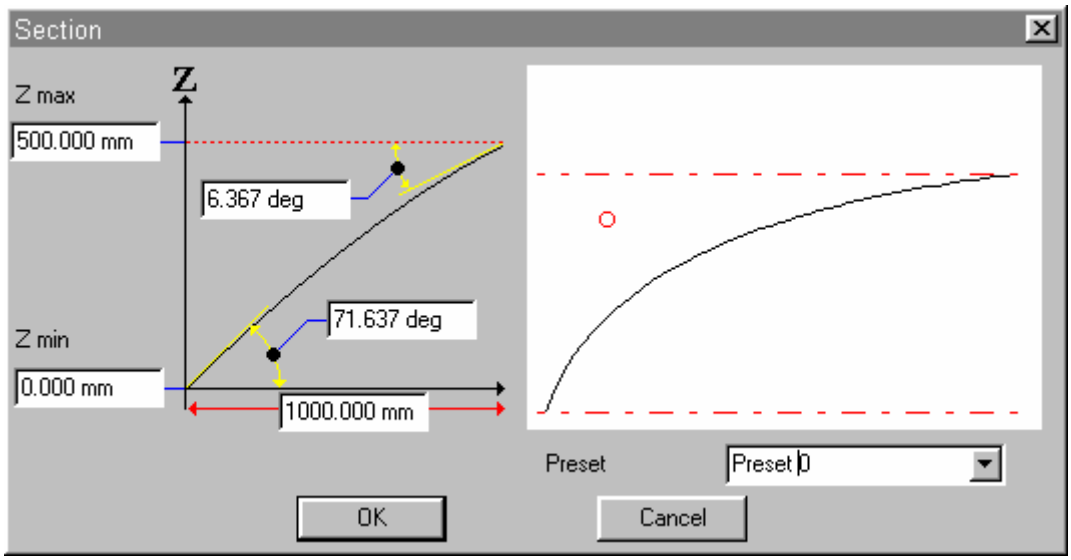
If your drawing goes outside the guidelines, go back in the CAD module by clicking on the Select **icon** **and resize the drawing.**

To go back again into TypeArt, select the square around your drawing (it becomes red) and click the **TypeArt** icon .

If your square is not selected, the window for creating a TypeArt object is displayed again. Click on **Cancel** since you do not want to create a second TypeArt object, and re-select the square.

Choosing a predefined section

Select only the grapes and bole, and click on the **Section** icon .
Parameters for the sections appear in the following dialog box:



Several shapes are already predefined.

For this example, chose shape number 5 and click on **OK**.

Next, select only the external contours of the leaves. Then click on the **Section** icon and choose shape number 2.

Select the Subtract combination mode for merging sections.

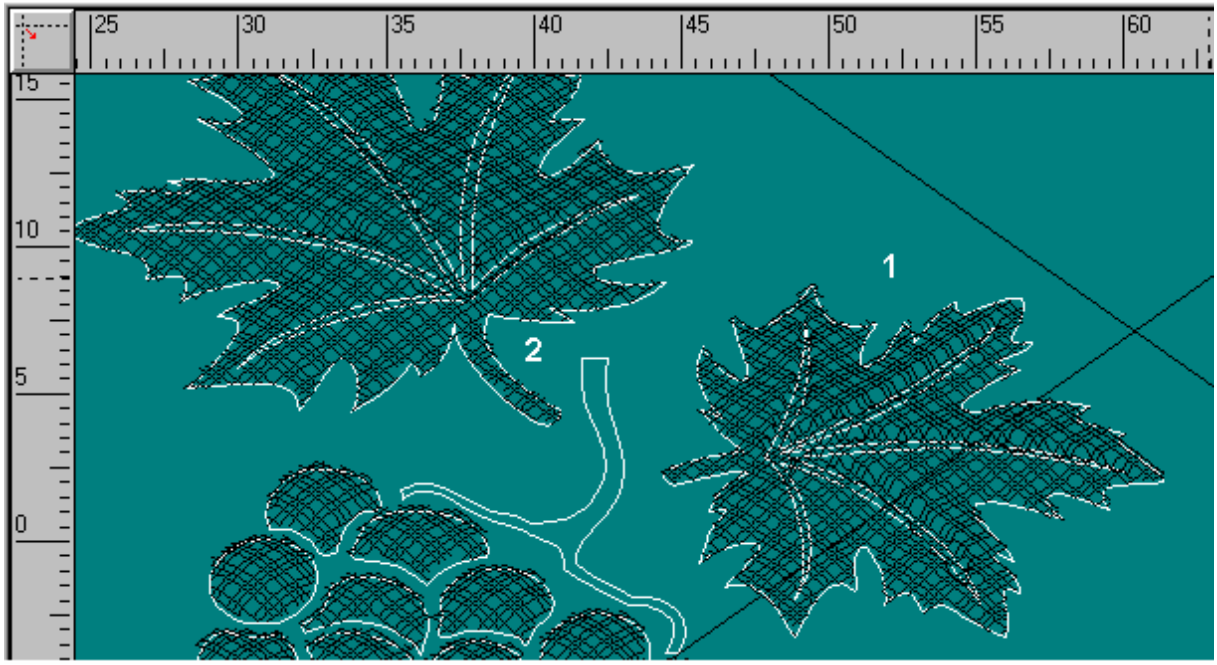
Select the vein of the right leaf and manually enter the following parameters:

Z max:	0,4 mm
Angle end:	0,0°
Z start:	0,3 mm
Angle start:	45°
Distance max:	0,6 mm

Click on **OK** when done.

Select the other vein and enter the following parameters:

Z max:	0,4 mm
Angle end:	0,0°
Z start:	0,0 mm
Angle start:	45°
Distance max:	0,6 mm



You will notice that the veins of the first leaf (1) with a Z start at 0.3 mm inside the existing section descend much lower than the ones of the second (2).

Sending the TypeArt to machining

From **TypeArt** click on the **CAM module** .
In the CAM module select **3D ISO** view under the **View** menu.

Getting ready for the toolpath

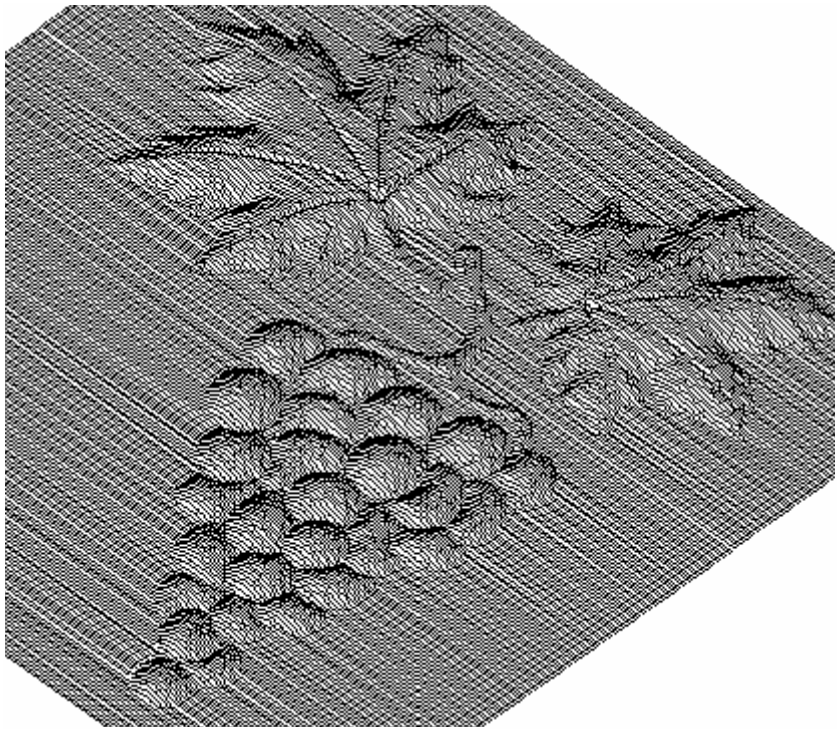
In the CAM module, select your composition and save it with a **.vnd** format using the command **Save as…** in the **File** menu. Click then on the icon **Create toolpath** . Two options for the creation of a TypeArt toolpath are available in the dialog box that is displayed on the screen: double-click on one of them. The dialog box for the creation of a TypeArt toolpath is displayed on the screen: click on the to select a tool. Enter then the following parameters:

Tool name: **Default**

Machining characteristics:

Overthickness: **0.000 mm**
Distance between steps: **0.150 mm**
Sweeping angle: **0.000°**
Machining mode: **Sweeping**

All these windows are described in the CAM section.



Tool path in 3D ISO view

Machining of the TypeArt object

Select the toolpath in the dialog box **Toolpath list**. Click on the mouse right button. In the context menu that is displayed on the screen, select the option **Machining…**.

In the corresponding **Machining** window select your machine, then choose the output: **Test**, **Port** or **File** and click on **Run** to launch the machining.

TypeArt: Basic tools

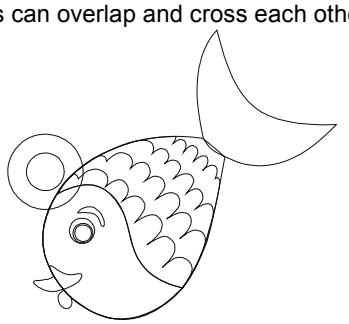
- ***The drawing***

The drawing is made under the **CAD module** with open or closed contours:

- either with classical tools: crayons, predefined objects (sketching tool, squares, circles), texts;
- from a vectorized image from a scanner;
- from imported files; or
- from all the creative possibilities of the **CAD module**.

Closed contours

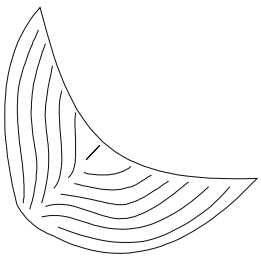
All geometries are allowed. The contours can overlap and cross each other.

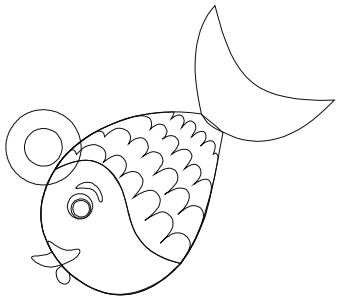


Open contours

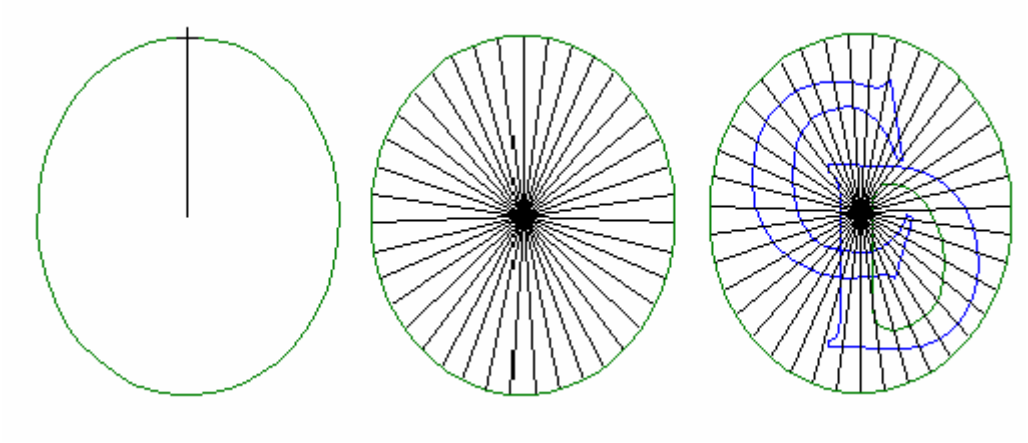
Open contours can be linked to a closed contour to create a raised effect or the bottom of an engraving.

Example 1: the tail of a fish





Example 2: special engraved background



Duplication of a line along a curve *Addition of letters*

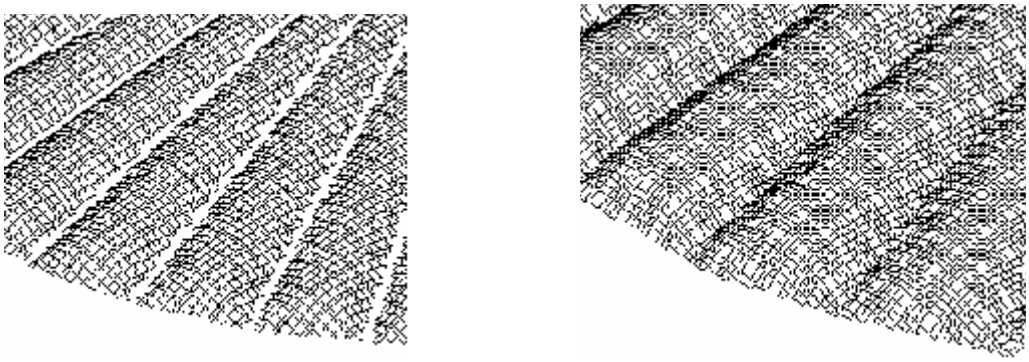
To use open contours, you must select them with one or several closed contours.

If you try to place a section using open contours, nothing will happen.

Remark

To avoid having a line that is too deep or holes instead of lines, which are at Z = 0 mm**, create first a small section on a closed contour, then select it with the lines and add volume.**

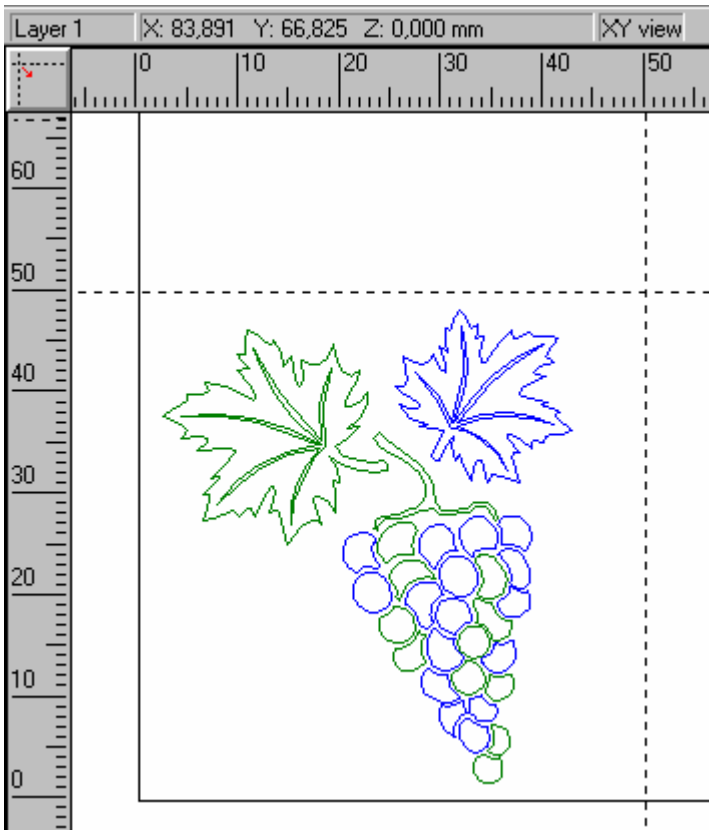
Example



Lines too deep *Right profile*

- ***Positioning the drawing***

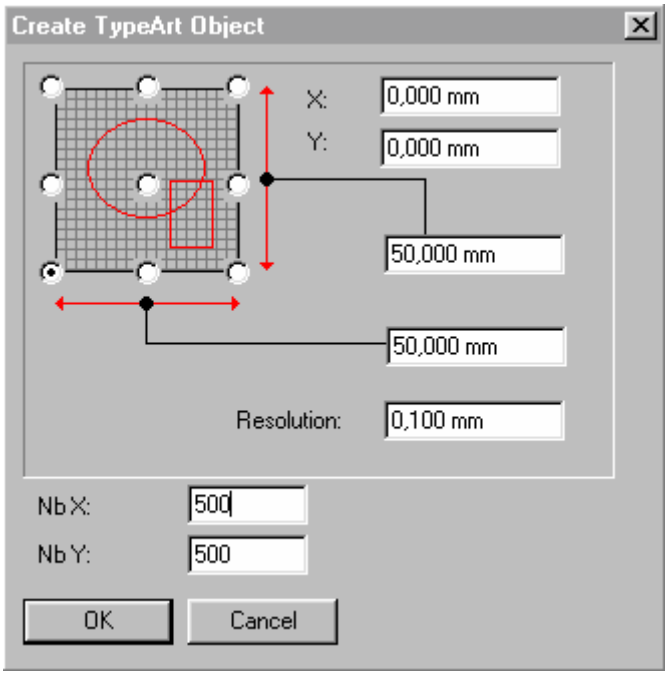
Once the drawing finished, give it the size you want and move it near reference point (X = 0, Y = 0) in the bottom left-hand corner of active area.



REMARK
Do not forget to save your artwork!

- ***Running TypeArt***

To run **TypeArt** click on its icon . The dialog box for creating a **TypeArt** object appears.

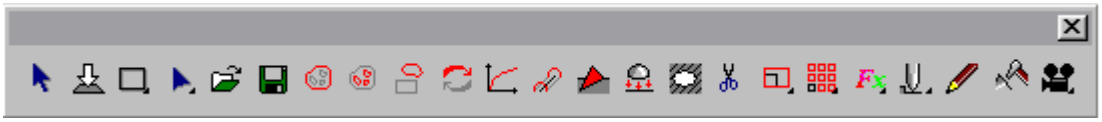


Box parameters

Nb X and Nb Y
These values correspond to the number of points desired on each axis.
The more points there are, the better will be the final result.
To make only virtual tests keep the default value of 200.
To get a better result in machining, choose a higher value (for instance 500 for an area of 50 x 50 mm that is to say a minimum step of 50 mm/500 = 0,100 mm in Sweeping mode).

X0 and Y0
Give the starting point of the box around your **TypeArt** object.
Default value: reference point (X = 0, Y = 0).

X1 and Y1
Give the size of the **TypeArt** box around your drawing.
Default value is 50 mm. You can create bigger or smaller boxes.
Adapt after the number of points on X and Y according to the size of the TypeArt object, keeping in mind that the larger it is, the more points will be needed to create a nice engraving.
The **TypeArt** screen and tool box appear.



The drawing is automatically displayed in the last view used (initially 3D ISO).

Hollow or raised drawing?
We recommend building you 3D design under TypeArt in relief, even if in the end you will produce a hollow mold. The TypeArt tools are easier to use with a raised drawing. Before getting to machining, you can transform the relief into hollow, and adapt the position in Z.

2D rendering of a TypeArt
The rendering of TypeArt sections is visible in 2D view, a working mode sometimes more adapted than 3D.
In the CAD module, mark the option **Render TypeArt in 2D view** in the **Display** tab of the **Options** dialog (F10 key). The direction of the light on this rendering can be changed anytime with the **Light direction** tool (TypeArt visualization tools).

• *Lay down tool / Create TypeArt*

The tool in the permanent toolbar in TypeArt has two different functions:

- When a modeling tool is active, one click on the icon lays down the tool. The mouse pointer gets back to .

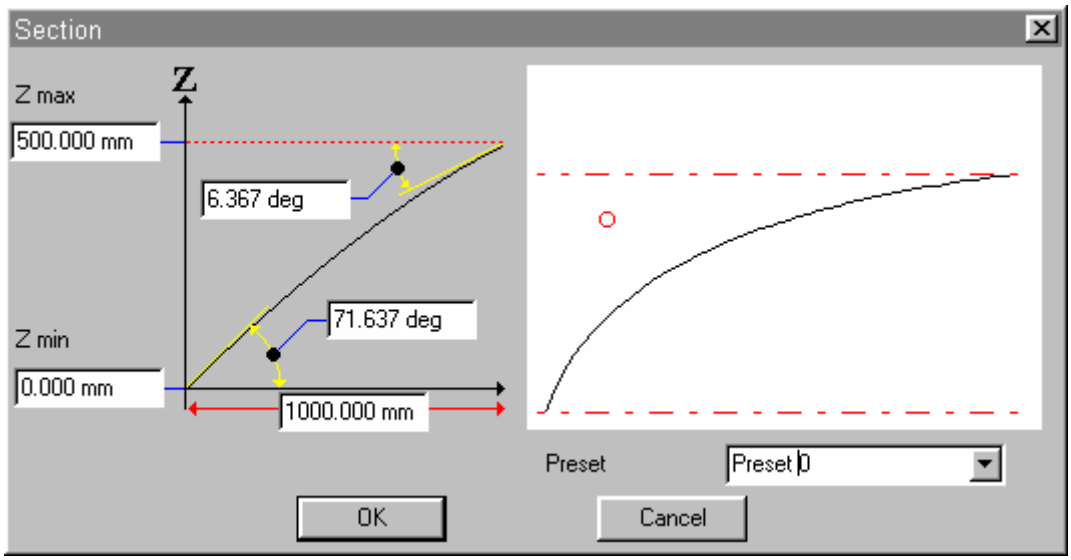
- When a shape tool is active, one click on the icon allows at the same time to lay down the tool AND transform the selected drawing into a TypeArt object. The mouse pointer gets back to .

To put down a shape tool, press the space bar.

• *Creating a symmetrical section*

Section
Select the contour on which you want to apply the section and click on the **Section** icon.

The **Section** dialog box will be displayed.



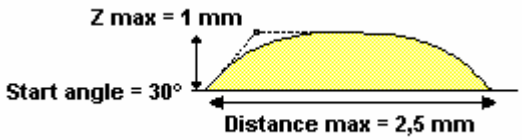
Parameters for defining the section

Z max
Maximum height of the section.

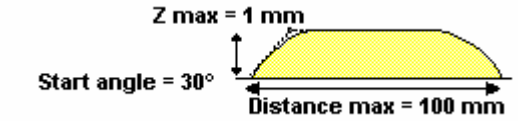
Remark

This limit will not be reached if the value in the Distance max **field is greater than the real distance between the contours.**

Distance max
This value defines and limits the length of the rise of the section.

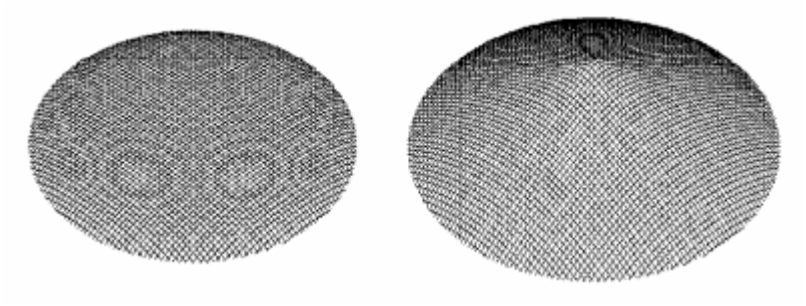


If this parameter is higher than the real distance between the contours, the shape will be flat at the top.



Ending angle
Gives the value of the angle on top of the shape.
Default value is 0. Negative angles are not allowed.

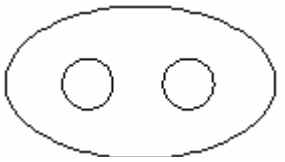
Examples



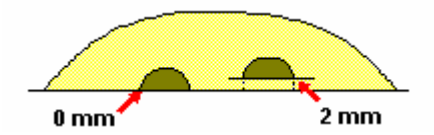
0° *45°*

This function allows you to design special sections or particular bottoms or tops of engraving.

Z start
Z of depart for the section. This function is useful to magnify details or extract a volume, especially in the case when merging several sections.



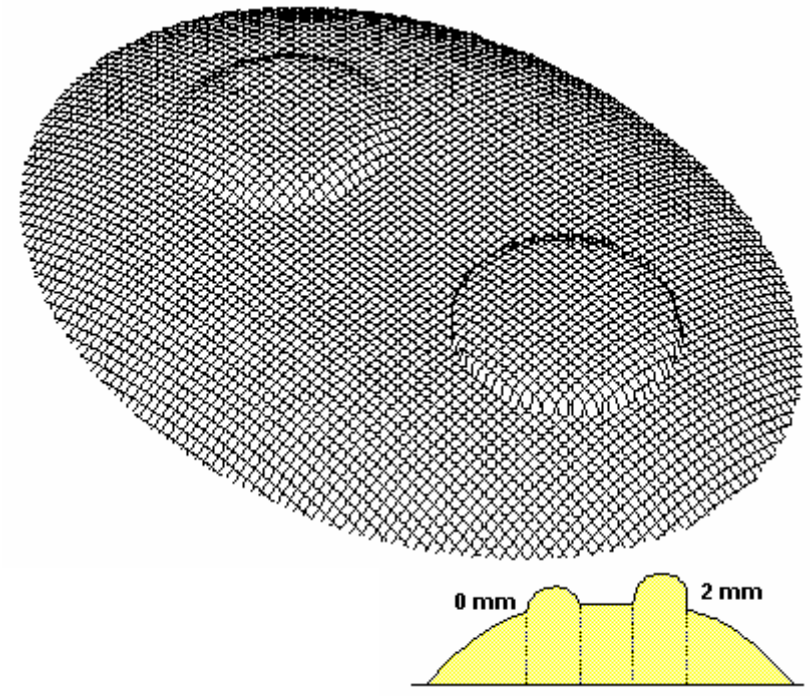
Drawing



First circle at Z start = 0 mm
Second circle at Z start = 2 mm

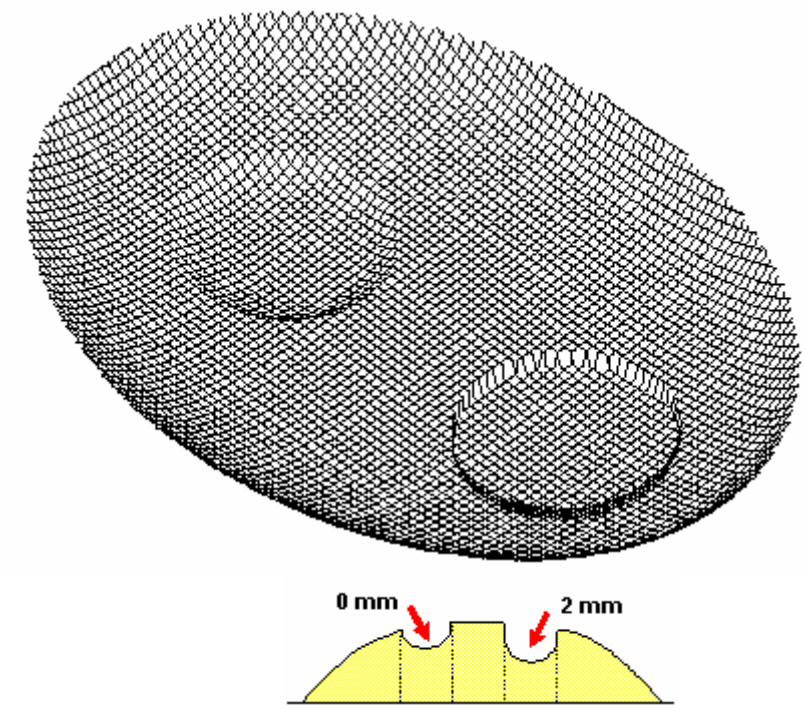
The value given in this field is added to the section whether in raised or recessed.

*The result with the **Add** option*



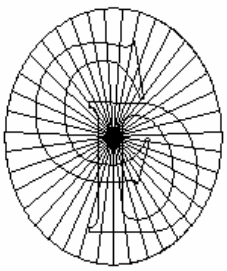
*The circle with **Z start = 2 mm** is much more accentuated (the excess material of 2 mm follows the initial shape).*

*Same result with the **Subtract** option*

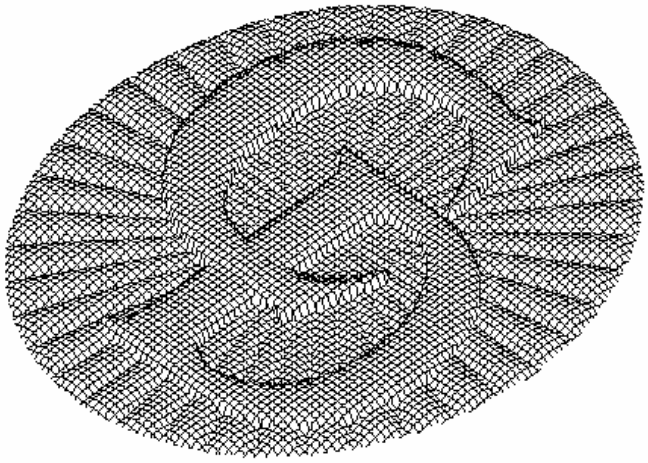


The same circle is also accentuated.

Sample with a special engraved background



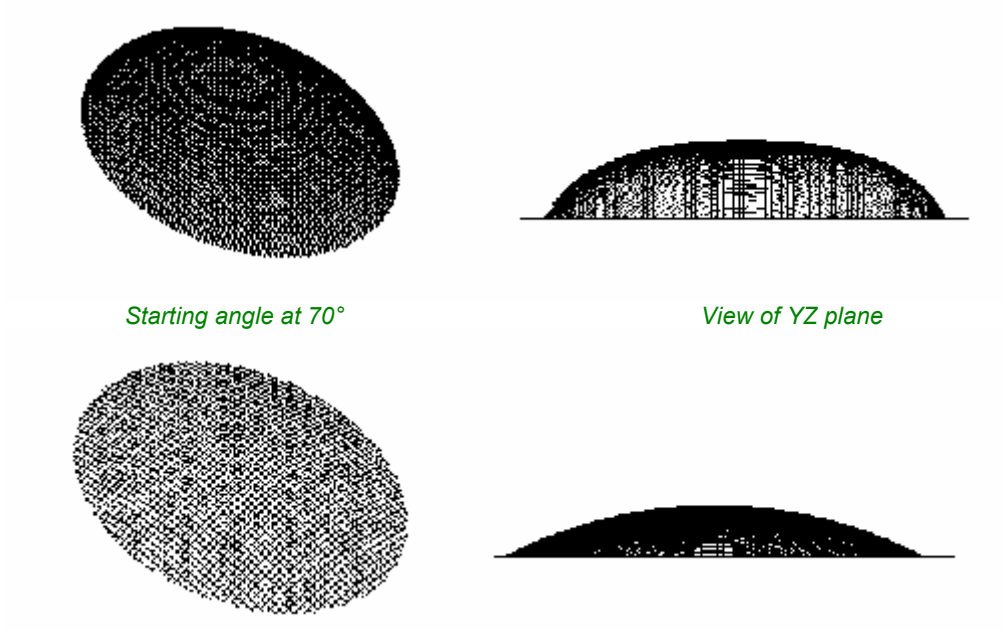
Drawing



Z start of 1 mm for the letter C

Starting angle
The starting angle of the section.

Example



Starting angle at 70° *View of YZ plane*

Starting angle at 25° *View of YZ plane*

Remark

To optimize the machining, select a starting angle that corresponds to the machining tool geometry.

Example for a tool with half angle of 30°
90° - 30° = 60°

You must give a starting angle less than or equal to 60°.

Bad angle *Good angle*

- ***Merging several sections***

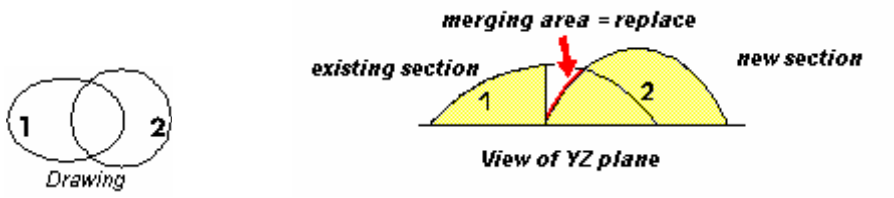
7 possibilities are available for merging or mixing sections on contours. They are shown in a floating toolbar you use before launching several operations on TypeArt objects:

Note : if the bar is not visible, it is probably reduced in the lower left corner of the screen. You might have to maximize the main window to see it.

Replace

All Z values of an existing section (Contour 1) will be replaced by the new ones (Contour 2).

Replace is the default option for the first shape.



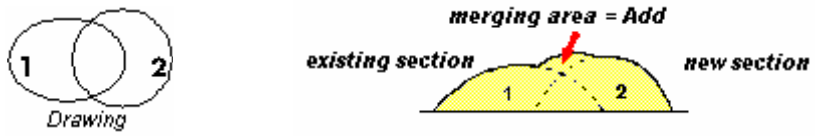
REMARK

The option Replace **is useful for deleting a section or an ensemble of sections without having to create a new TypeArt object.**

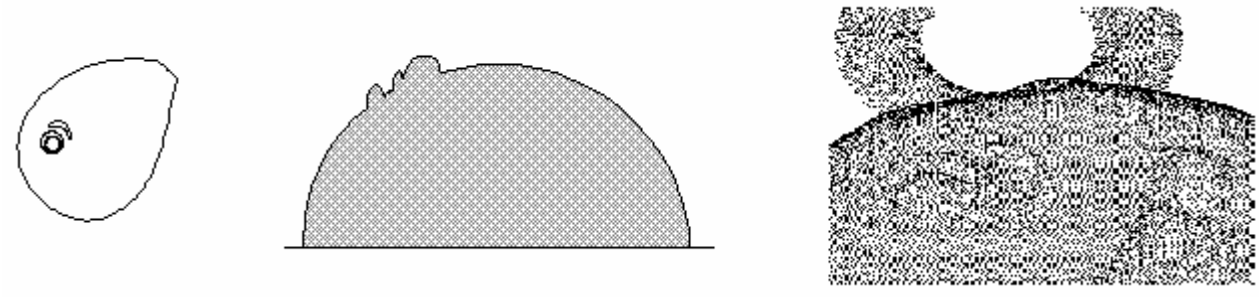
Adding

To add to an existing section (Contour 1) the entered value (Contour 2). In the merging area the two sections are added.

Example:



Example



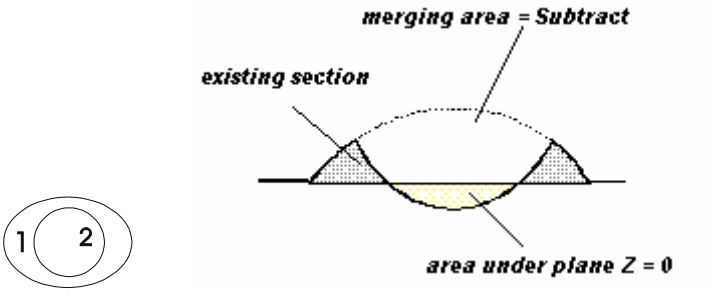
Drawing *View of YZ plane* *3D ISO view*

Subtract

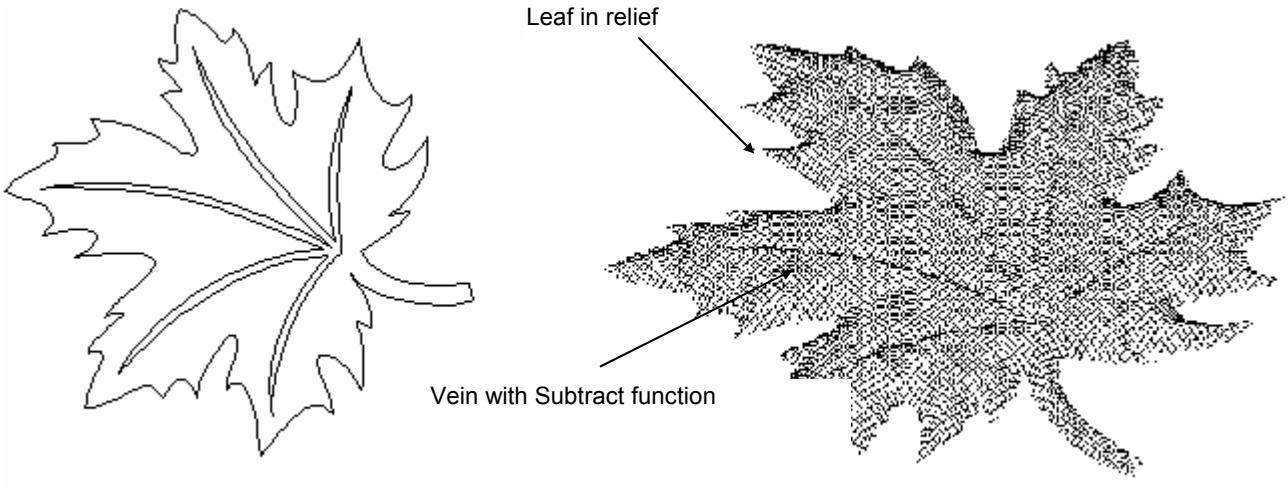
To remove from an existing section (Contour 1) the entered value (Contour 2).

REMARK

If the height of the subtracted area is higher than the height of the existing shape, you will get a negative value for Z in the merging area (see example).

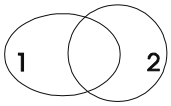


Drawing *View of YZ plane*

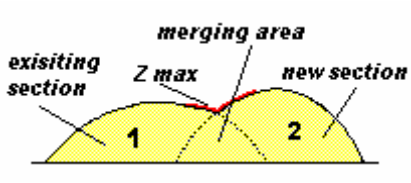


Keeping the highest section

Max (Z maximum)
The higher section will be kept.



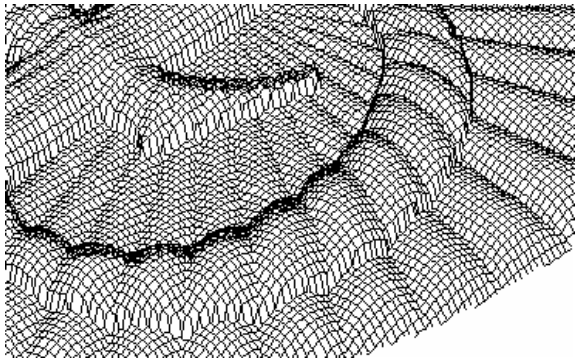
Drawing



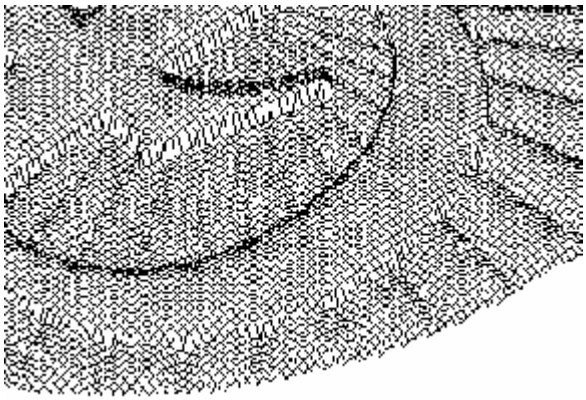
View in YZ plane

You must choose this function if the background of the engraving in relief is heavily accentuated in order to avoid reproducing it on above sections.

If you add the sections, the letters above the background have the same properties as the background itself. (Look also at the intersecting area between letters).



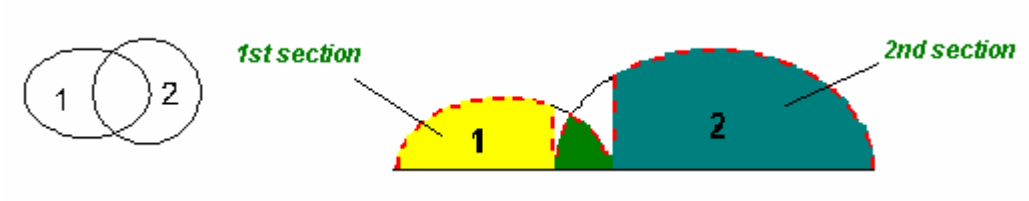
With the Z max option, the letters, as well as their intersecting areas, have their own defined and perfectly smooth sections.



Keeping the lowest section

Min (Z minimum)
The lowest section will be kept.

Example:



Drawing *View of YZ plane*

Keeping a middle section

Middle
In the common part, an intermediate section is kept in the middle of the existing sections.

Modification on the defined zone

Allows to apply a section modification only to the TypeArt area that already had a section. This mode will then change the application range of the other modes.

unactivated icon (no effect) activated icon (limitating effect)

• *Transforming a surface into a TypeArt object*

Any surface imported or created in the CAD module can be directly transformed into a **TypeArt** object.

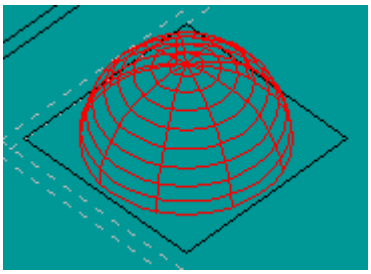
In the CAD module, select the surface to be transformed into a TypeArt object (you should better position it in the bottom left corner of the active area) and click on the **TypeArt** icon to switch to its module. Click then on the icon **Combine surface** . The current **Combination mode** applies (See section **Merging several sections**).

Select the desired option and click on **OK** to validate.
The surface is transformed into a TypeArt object.

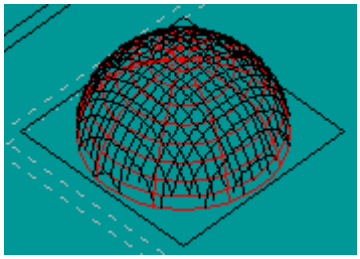
Remark

Some kind of surfaces like IGES **may generate imperfections (like** *holes* **in the meshed surface) during their transformation. Hit the** Shift **key meanwhile you click on the icon** Combine surfaces **to correct this. This operation will need a longer treatment.**

Example



3D surface



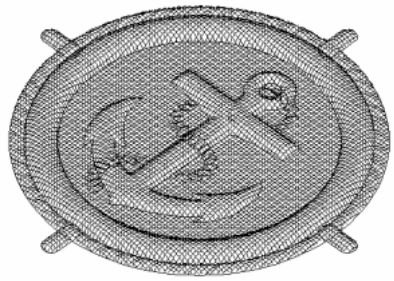
Transformation into TypeArt

- ***Erasing part of a TypeArt area***

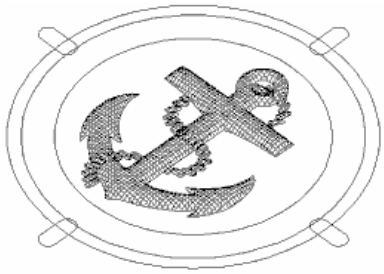
This feature allows to erase particular areas of a TypeArt object while keeping remaining areas.
First of all, select an area of a TypeArt object.

Click now on the **Erase TypeArt area** icon. Only the selected area will be kept. What remains is erased.

Example



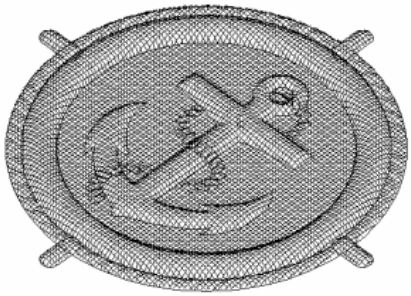
Select only the anchor



Click on the icon

You can also push the **Ctrl** key while clicking on the **Erase TypeArt area** icon. The selected area is erased. Remaining areas will be kept.

Example



Select only the anchor

*Press **Ctrl** and click on the icon*

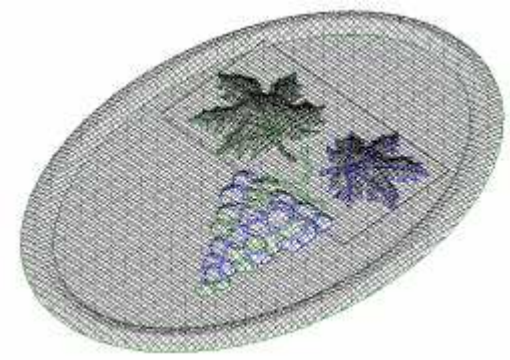
- ***Extracting an area from a TypeArt object***

This feature allows you to cut and paste an area from a TypeArt object in order to create a new object using the extracted area. This feature is especially useful to get back specific parts from a TypeArt object to embed them in another TypeArt object.

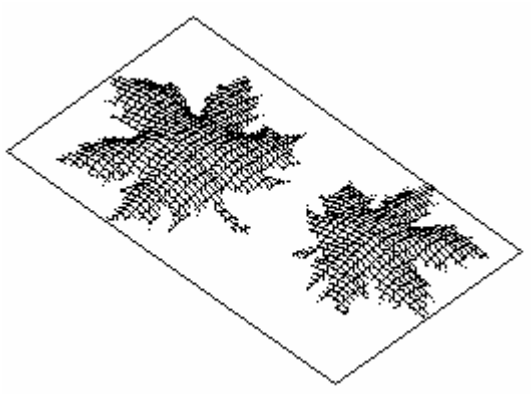
First select the part to be extracted. Click then on the **Extract TypeArt area** icon. Place your pointer on the area to be extracted and click on the left button of your mouse. Keep the mouse button pushed and drag your pointer until the desired point. Release then the mouse button. The TypeArt area has been extracted and

displaced.

Example



First select the leaves



Drag then selected area

- ***Creating a non symmetrical section with the aid of markers***

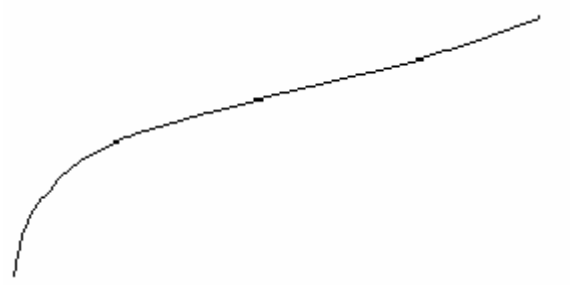
Principle

The principle consists of elevating one part of a section to a predefined value in Z with one or several markers.
The profile, i.e. the starting / ending angle, is defined by a curve: segment, arc, etc.
It is either drawn with the **Sketching tool**, imported from a plane, or created in the **CAD module**.

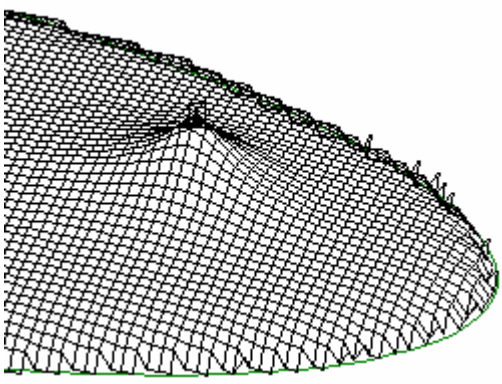
Geometrical profile

The profile is defined on the XY plane of the page without any restraint regarding position or direction. Only its geometry is essential in determining the form of the section.

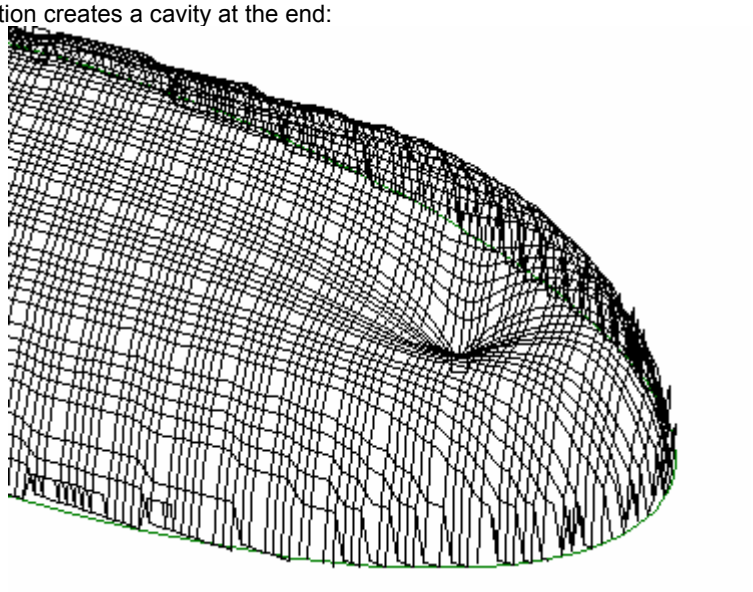
With a rising profile

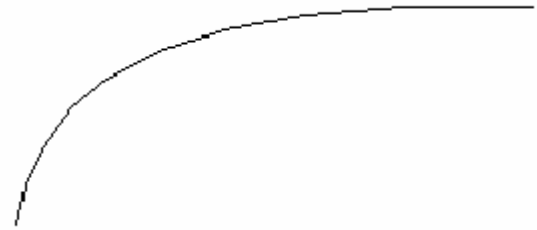


The section climbs to an angle corresponding to the ending angle of the profile

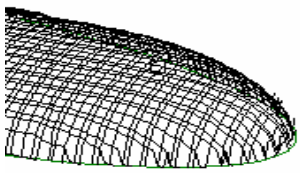


With a descending profile
The section creates a cavity at the end:





With a leveled profile



The section is leveled out at the end:

Example

Draw an ellipse with a length of 45 mm.

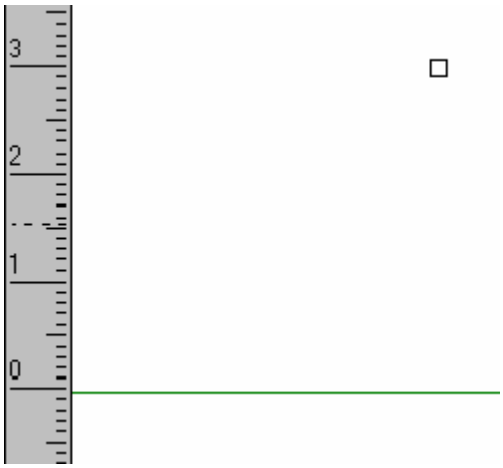
Place a marker inside the ellipse on the right side using the mouse or the **F2** key.

Convert the marker into a curve.

With the shaping tool, double-click on the marker which becomes encompassed by a square. The dialog box **Points attributes** is displayed. Enter a value of Z = 3, and validate.

You can also place the marker with the **F2** key.

Check in the YZ view that the point is accurately located at 3 mm.



- ***Creating a constant height section with the aid of a curve***

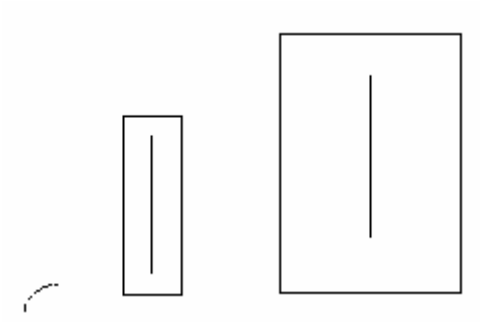
In the simple functions of **TypeArt**, the height of the embossment depends on the Z max (height) and Max Distance (distance between each contour) parameters. It is therefore difficult to obtain a section with a constant height when for instance a drawing is thinner at some extremities.

Principle

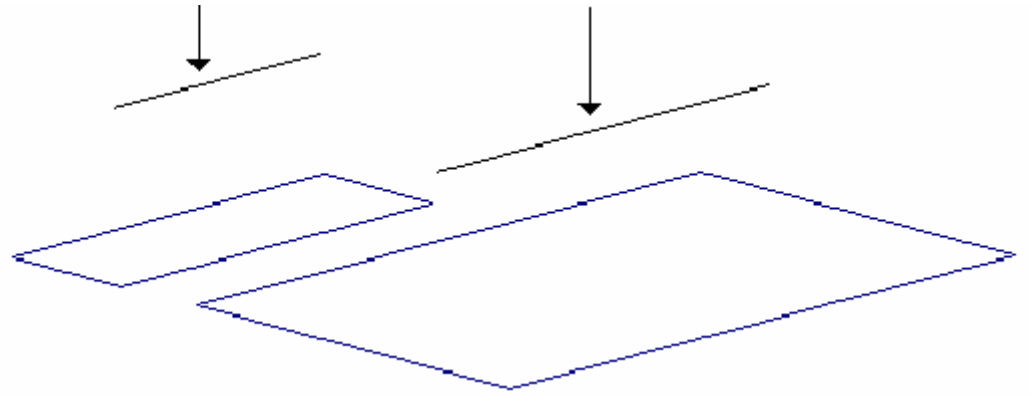
It consists in limiting the maximum height of the section by a curve whose points of construction have a preset value in Z.

The starting and ending angle varies depending on the height of the section (limited by the points of the curve) and the distance between the contours. Its shape is defined by a tracing: the profile.

In this drawing, the construction points of both median curves are in Z = 5 mm.

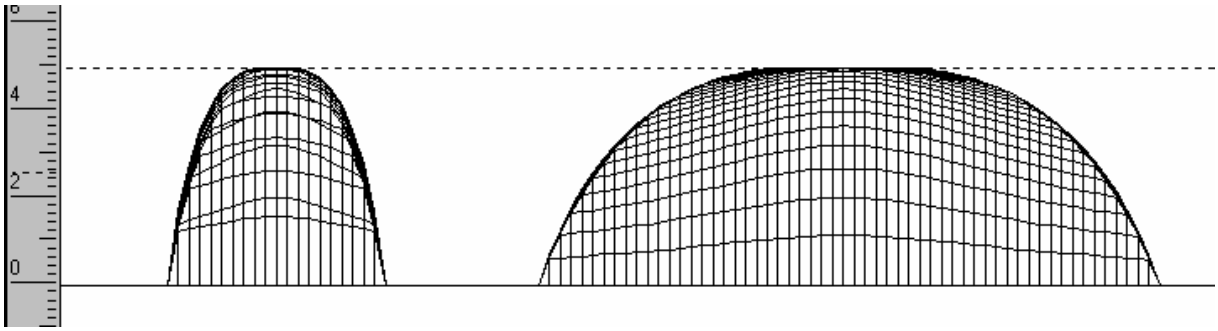


The profile is an arc leveled out at the end.



Embossments obtained in **TypeArt** for each of the drawings in XZ view.
The starting and ending angles are different.

Different starting/ending angles *Z constant at 5 mm*



The geometry of the starting and ending angles are changing according to the distance between the contours.
To obtain a starting/ending angle identical to the profile ones, the length of the profile should be equal to half of the distance between the contours. This applies when the distance of a contour is constant.

Example with a drawing

Type the number **0** on your screen and select a font, such as **Times New Roman Bold**, in order to have a variable distance between the contours and to illustrate better our example.

Give the number 0, a height of about 50 mm and position it at the lower left corner of your screen.

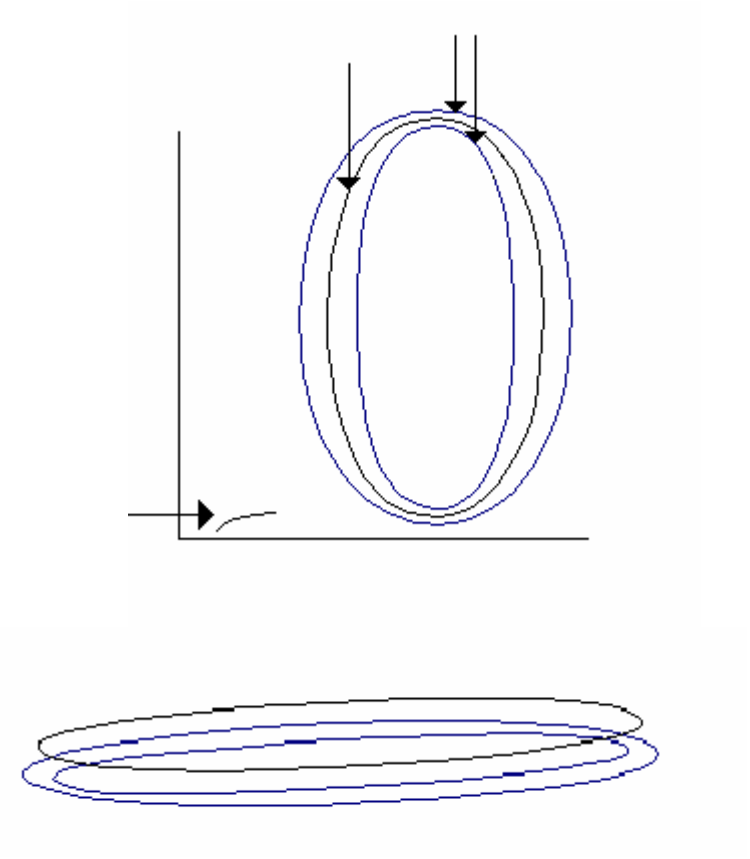
Trace the median curve to the contours with the **Sketching tool** and *be careful to leave the contour open*. (For more convenience, close the curve on the last point and then use in the shaping tool box the **Disconnect** icon to open the curve at this point).

Trace the profile of the section.

For practical reasons, we advise you to position the profile close to the drawing.

Select the median curve and drag it with the mouse. Without deselecting the object, press the F2 key.
In the window **Move to**, enter the following values: **X = 0 mm**
Y = 0 mm
Z = 4 mm

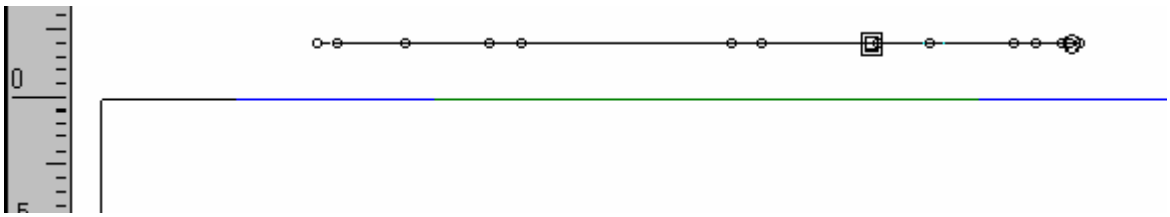
Median contours of 0



3D view

The median curve is an open tracing (black contour), the contours of the number 0 are closed tracings (in green or blue).

Check in XZ view that all the points are located at 4 mm and then return to XY view.



You can now run **TypeArt** by clicking on the icon .

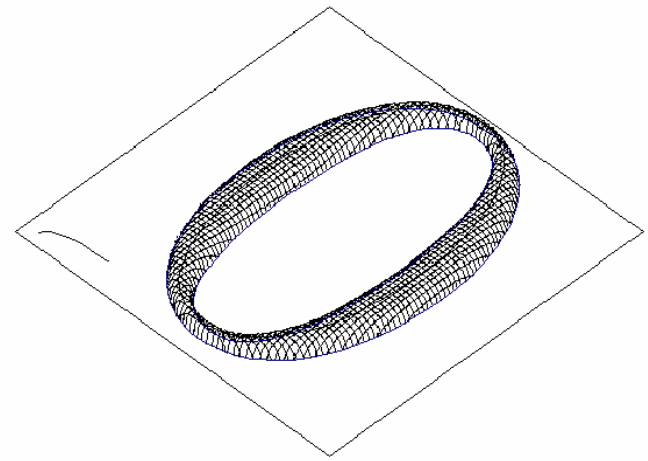
Enter the following parameters:

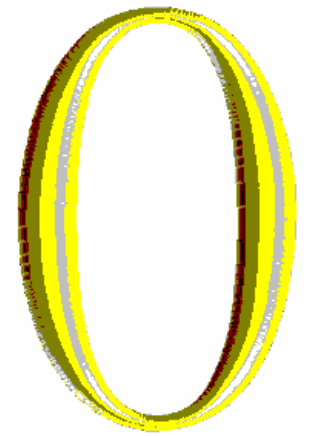
Nb X = 400
Nb Y = 600
X0 = 0
Y0 = 0
X1 = 40
Y1 = 60

Remark

Check that the drawing fits into the TypeArt **box.**
Select the combined contours of 0, then the median curve (opened), and finally the profile by using the **Ctrl** key for multiple selection.

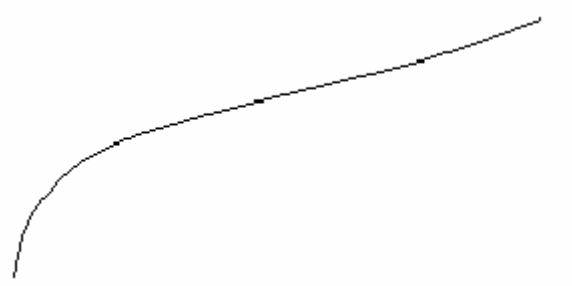
Press the **Ctrl** key and click on the **Section** icon .





Remark

If the curve with the Z max is visible (a cavity or a vertex), it means that your profile is not leveled out at the end.



the shape will climb at an angle and create a vertex

the shape will create a cavity

In that case, modify the profile and redo the TypeArt.

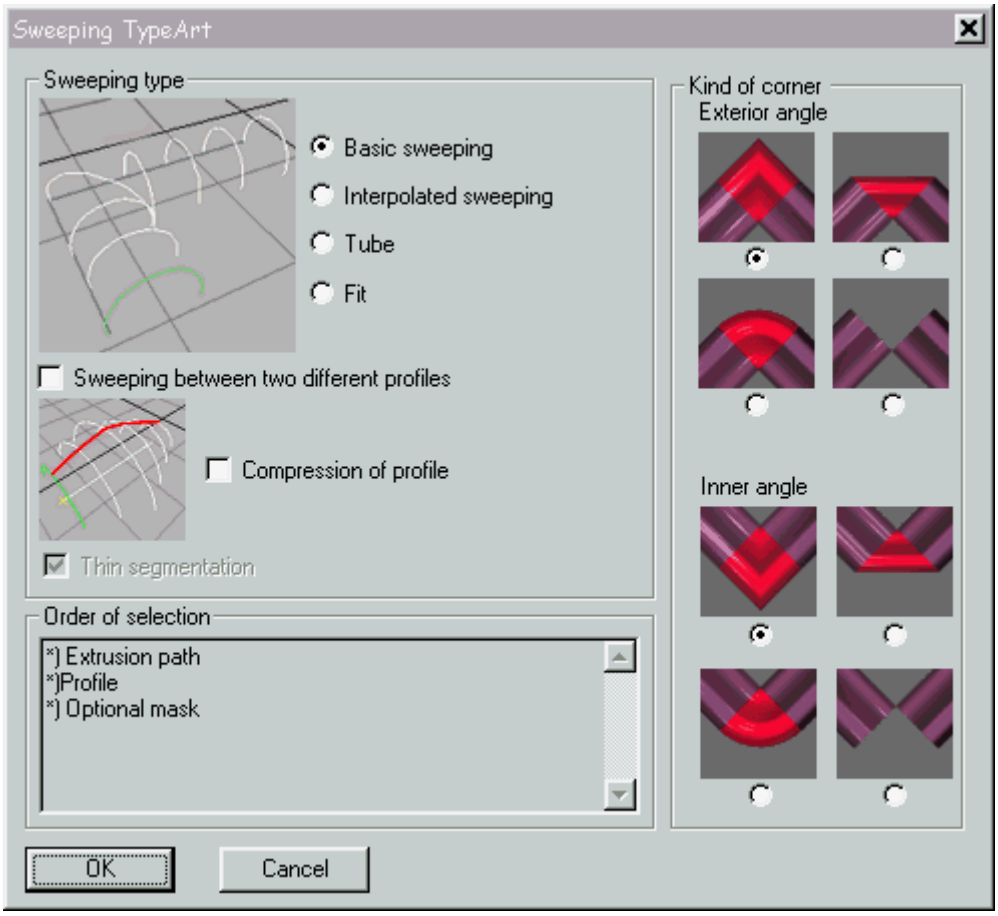
- ***Curve sweeping***

The **TypeArt** module includes a powerful function allowing you to sweep (some may also say *extrude*…) a shape on one or several curves. Four kinds of sweeping can be done using this function.

1. Basic sweeping

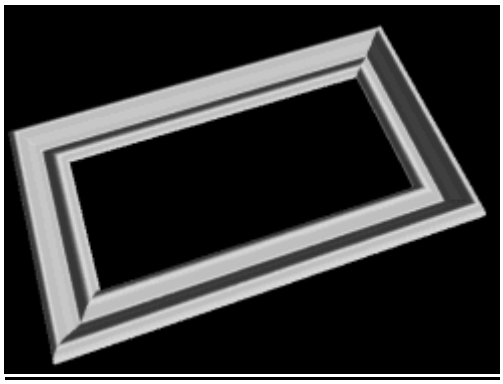
This function is specially meant for designing frames, as the following example shows.
Create a profile shape (the molding of the frame) and a rectangle (the support curve, that we may call a section curve, and on which you will *sweep* your profile shape).
Note: if the contour is clockwise, the shape is generated on the outside; if the contour is anti-clockwise, it is generated on the inside.

Select both objects and click on the **TypeArt** icon . In the **TypeArt** module, select the section curve first and then the profile shape. Click then on the **Curves sweeping** icon while pressing the **Ctrl** key. The following dialog box appears:



Validate with **OK**. The current **Combination mode** applies. TypeArt fuses the profile through its leftmost point and the section curve. The shape is created on the inside or outside of the section curve depending on the startpoint of the section.

Support curve *Profile shape*



Curve sweeping (Viewer TypeArt)

REMARK

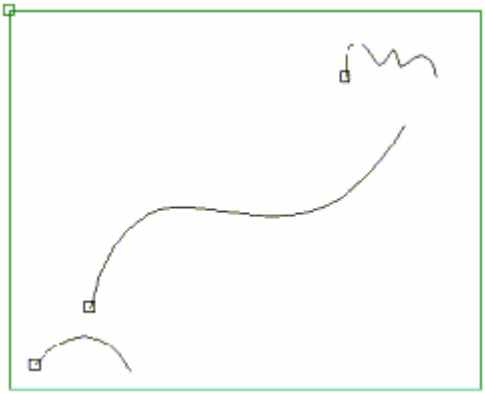
In the dialog box Sweeping TypeArt **you have an active control on the way the angles will be generated. When the angles of the initial curve are to sharp, it becomes impossible to create a shape on these angles. The solution is to** *open* **a little more the angles or to round them in order to create a profiled shape.**

1a. Basic sweeping with two profiles

In the **CAD** module, create two profile shapes and one support curve. The geometry of the resulting section is determined by the sense and the selection order of the profiles.

Select those three objects, and click on the **TypeArt** icon . In the **TypeArt** module, select the support curve first, then the profile chosen for the starting point of the curve, and finally the profile taken at the other end of the curve. Click on the **Curves sweeping** icon while pressing the **Ctrl** key. Select **Basic sweeping** in the dialog box that appears, marking also the **Sweeping between two different profiles** box**.**

You can use a compression curve to limit the height of the sections, or to produce special effects. In that case mark also the **Compression of profile** box. The shape is then created taking into account the compression curve (the last selected) for its height.



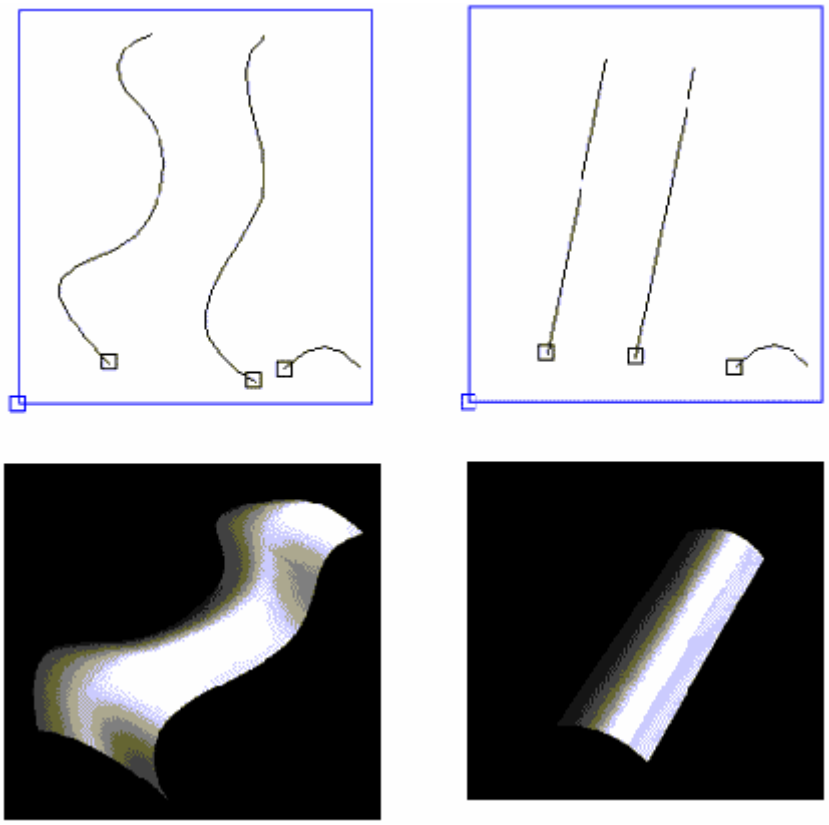
Profile 1 Support curve Profile 2

Sweeping (Viewer TypeArt)

2. Sweeping between two section curves

In the **CAD** module, create one shape profile and two section curves (or paths). Both paths must have the same direction, or TypeArt will generate a knotted curve.

In TypeArt, select first the paths then the profile. Click on the **Curves sweeping** icon while pressing the **Ctrl** key. Activate the option **Interpolated sweeping**. You can also use a compression profile. Click on **OK** to validate. The current **Combination mode** applies. TypeArt links the two sections with the shape profile.



Examples of sweeping with two paths

2a. Sweeping between two curves using two profiles (*Morphing* effect)

In the **CAD** module, create a start profile, an end profile and two section curves. Select the four elements and click on the **TypeArt** icon . In the **TypeArt** module, select first the section curves, the start profile and finally the end profile. Click on the **Curves sweeping** icon meanwhile you strike the **Ctrl** key. In the dialog box Sweeping TypeArt that is displayed on the screen, activate the options **Interpolated sweeping** and **Sweeping between two different profiles**. Select angle options if needed. You can also make use of a compression profile. Click on **OK**. In the displayed dialog box, select the desired combination mode and validate through **OK**. TypeArt makes a morphing effect between the two section curves.

Sections 1 and 2

Start profile 3

End profile 4

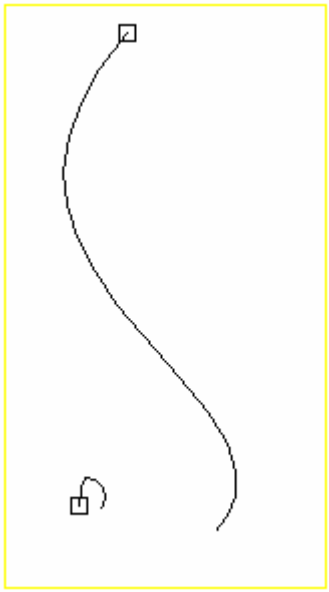
Sweeping with morphing

3. Tube Sweeping

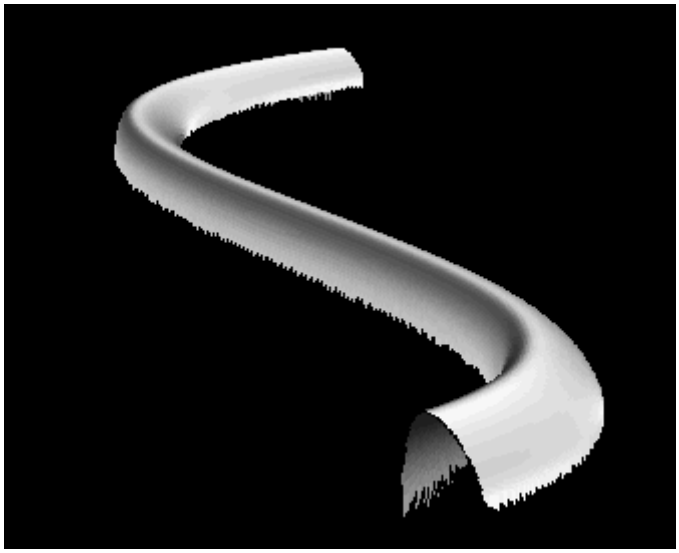
Allows to define a tube that will have the winding of the path, and the smooth shape of the profile all along.

In TypeArt, select the path then the profile. Click on the **Curves sweeping** icon meanwhile you strike the

Ctrl key. Select **Tube** in the dialog box that appears. Click on **OK** to validate. The current **Combination mode** applies.

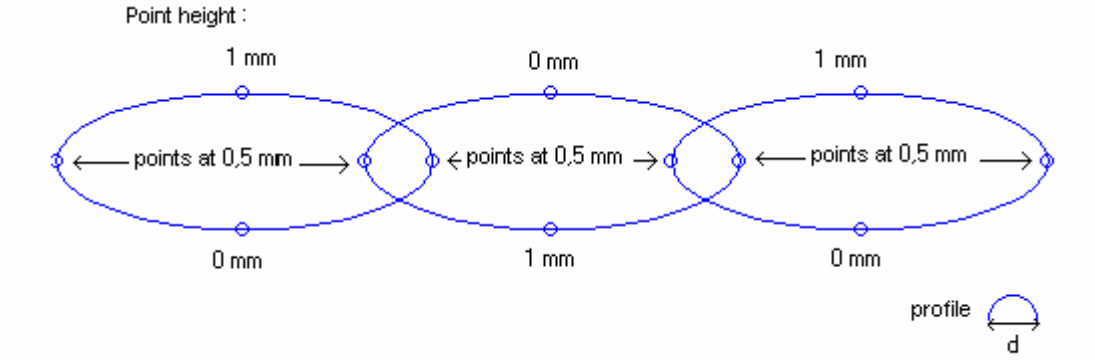


Original path and profile

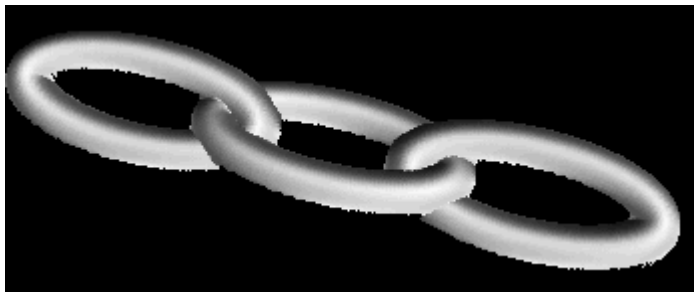


Resulting Tube sweeping (TypeArt Viewer)

Example producing the links of a chain:
Build the chain with three ellipses having points at different heights, and a profile in half-circle shape.



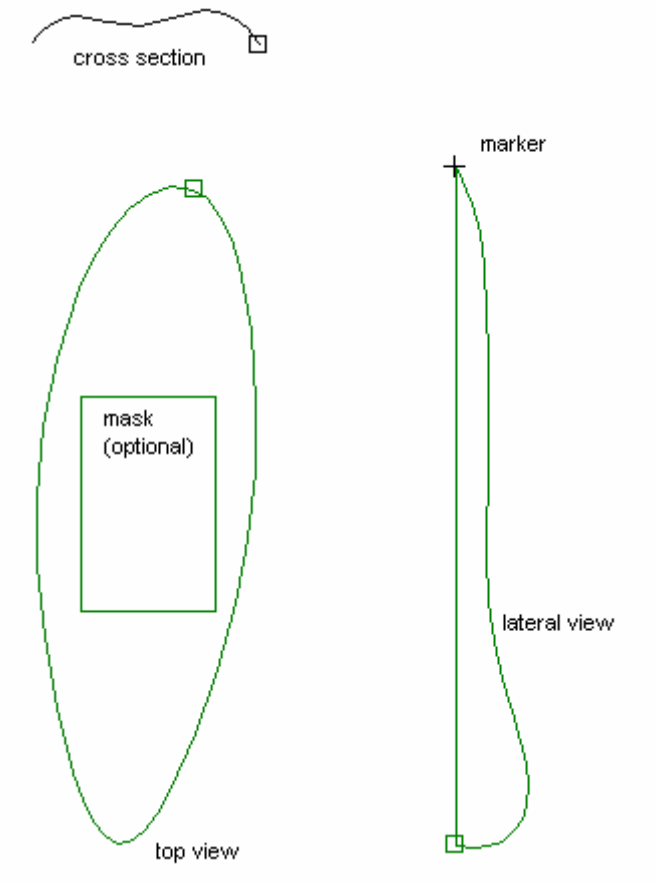
Select the first ellipse then the profile, and click on the **Curves sweeping** icon**.**
Chose the Combination mode **Max** to intertwine the rings properly. Do the same for each ellipse.



4. Fit sweeping

For this complex type of sweeping, you need to be very careful in managing the views.

Building the petal of a flower.

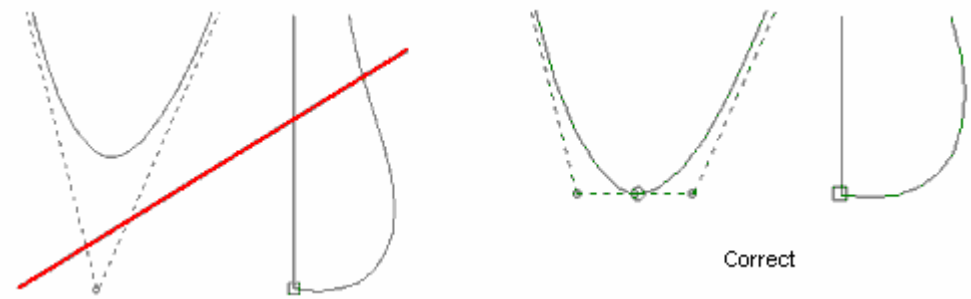


Cross section: it is the profile of the section, as a cross section (open or closed contour).

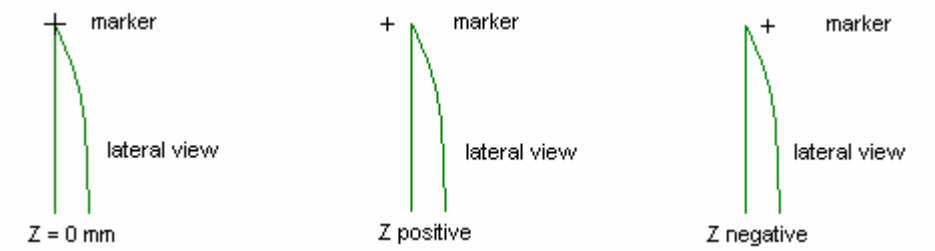
Top view: the sweeping is done inside this contour (section) that must be closed. The **mask** is used if you need only a part of the area to limit this portion.

Lateral view: it is the profile of the section, seen from the side. The contour must be closed.

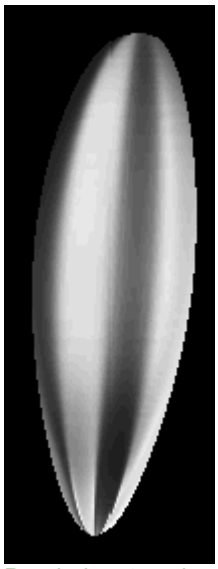
Warning: top view and side view must have exactly the same dimensions, and be aligned. Use the vertical auto-dimension and alignment tools and pay attention to align the contours themselves, and not their control points.



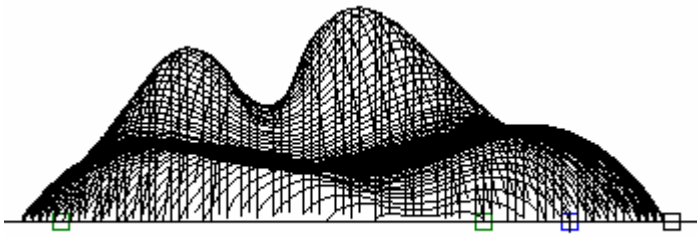
The marker: situated on the lateral view, it is the reference point of the section that determines the Z starting.



Selection order of the contours: select first the top view, then the lateral view, the marker, the cross section (profile), and finally the mask if needed.

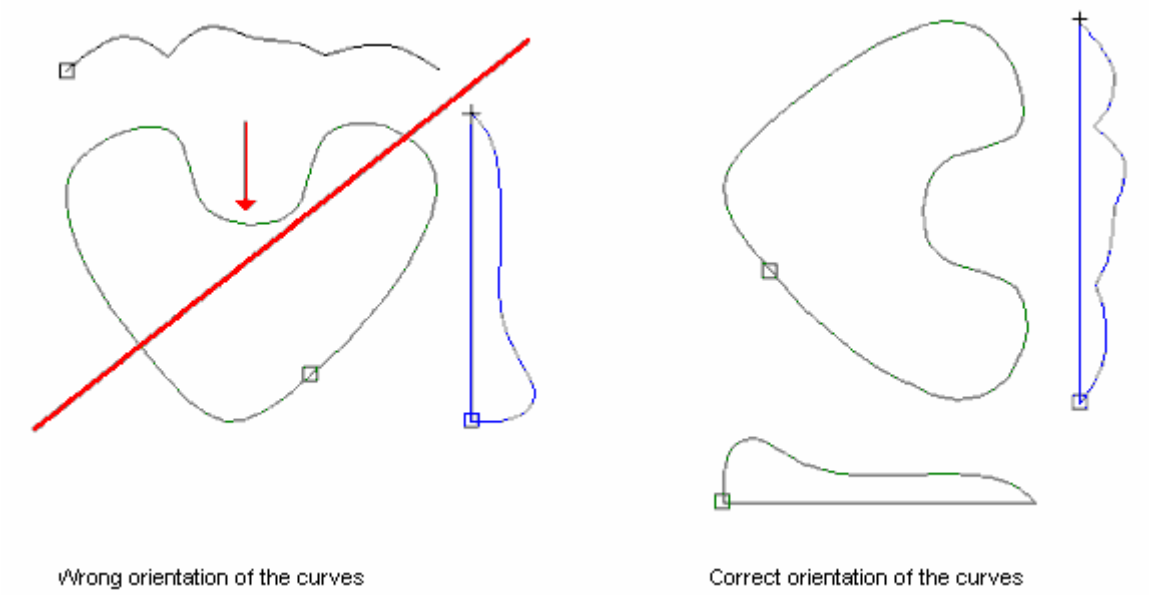


Rendering, top view



Rendering, side view

Note: when building the view above, you must avoid inward shapes on the X axis, because they could generate an error message. For example:



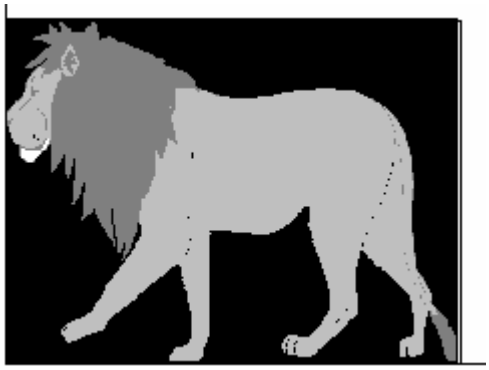
Wrong orientation of the curves

Correct orientation of the curves

- ***Creating a section from a scanned picture***

Scanner
Place the picture in the scanner and click on the scanning icon.
Choose the appropriate vectorizing parameters and make sure to select gray levels for the picture.

Example: Type: Photo in gray levels.



- ***Importing the picture from a Bmp or Tiff file.***

Import
Click on the **Import** icon and from the directory select the file with the Bmp format from the directory.

Remark

To display the picture on the screen, validate the option in Edit→ Options… → Show Bitmap**.**

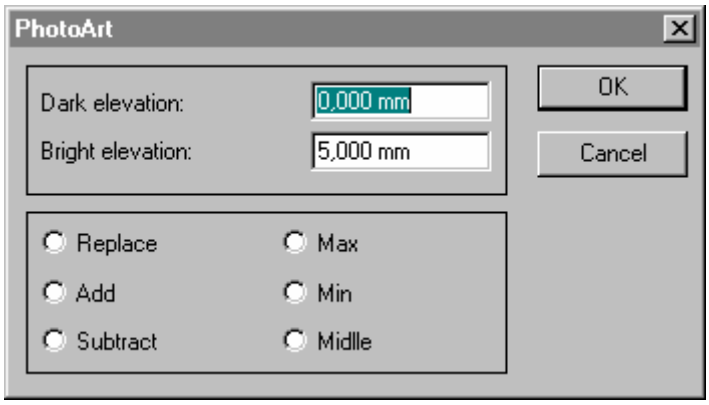
Positioning the picture

Resize the picture to the required format and position it preferably in the lower left-hand corner, origin X0 Y0.

Running TypeArt

Select the picture by clicking on the external frame and then click on the TypeArt icon.

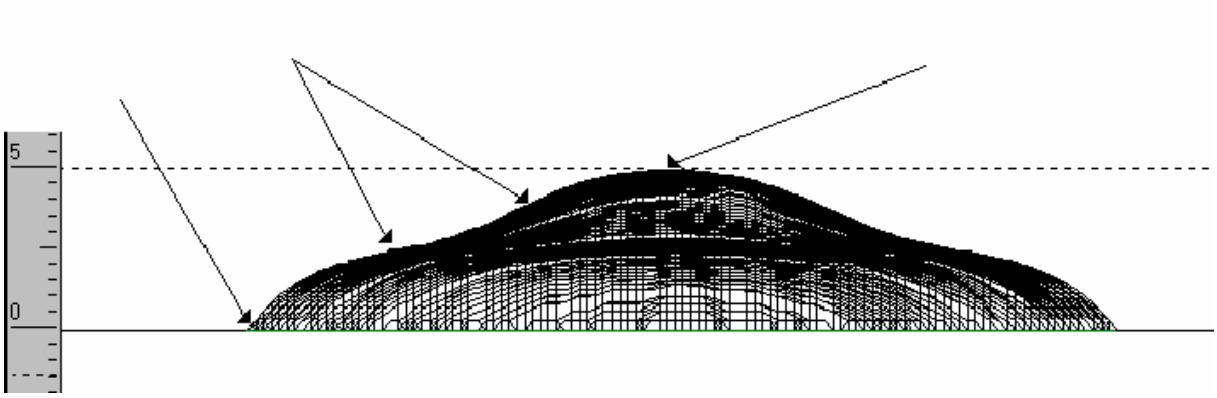
Automatic elevation



Dark elevation
Value of the maximum altitude of the darkest gray level, in other words, the black area.

Bright elevation
Value of the maximum altitude of the brightest gray level, in other words, the white area.

Example
Dark elevation = 0 mm
Bright elevation = 5 mm



Elevation with a parameterized profile

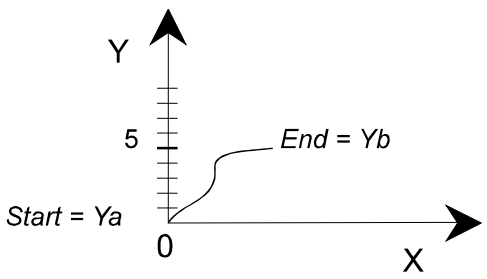
The parameters of automatic elevation given in the **PhotoArt** window (linear variation or proportional) can be replaced by any curve: the profile.

This profile must be drawn as explained below.

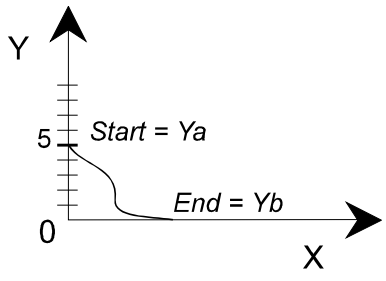
Starting point
The starting point of the profile (Ya value) determines the elevation of the darkest part of the drawing: the black. Its position has to be defined in accordance with the picture and the desired result (in recessed or in relief).

End point
The end point of the profile (Yb value) determines the maximum elevation of the shape.
It corresponds to the clearest gray level: the white.

Example



The altitude of the white part of the drawing is 5 mm in Z.
Ya = 0 mm Yb = 5 mm



The altitude of the white part of the drawing is 0 mm in Z.
Ya = 5 mm Yb = 0 mm

Construction of the section

Select the picture (by clicking on its frame) and then the profile. Next click on the **TypeArt** icon .

TypeArt is automatically executed. The resolution of the scanned picture will define the resolution of the TypeArt object (number of X and Y points initially entered in the TypeArt object creation box).

Correcting sections

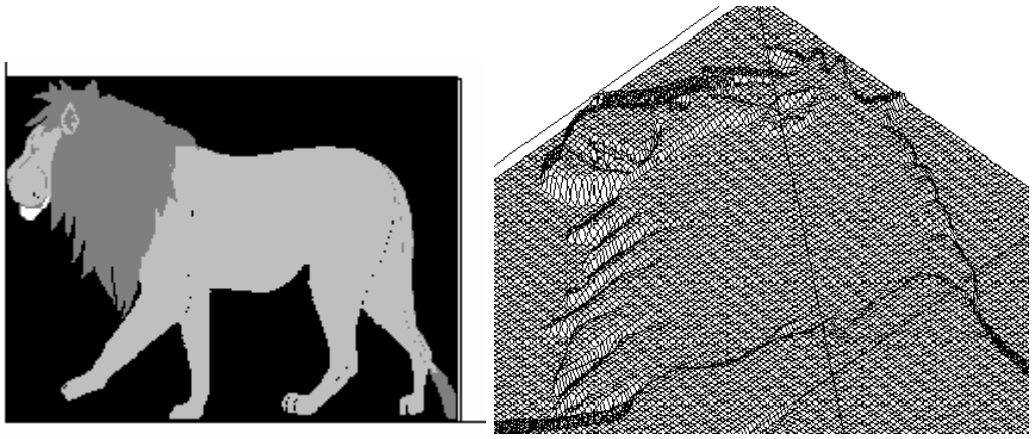
In some compositions, the gray levels are not adapted for relief.
For instance, the dark parts (shadows, colors, etc.), which give a volume effect to flat pictures, must be in relief on a 3D object.

It is possible to correct these shapes.
In the CAD module (in XY view) redraw the contour of the area to be raised with the sketching tool.
Then in TypeArt add an embossment onto the already existing one.

Example

According to the principle that dark parts are lower than bright parts, then the mane of the lion will be recessed, in other words, less elevated compared to the body of the lion (bright parts).

Select the picture, click on the **TypeArt** icon and enter the following parameters:
Dark elevation = 0 mm
Bright elevation = 3 mm



In the CAD module **XY view**, select the **Sketching tool** and redraw the contour of the mane.

Sketching of the mane

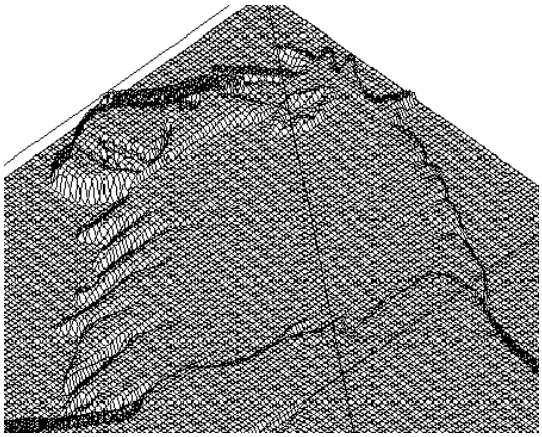
Remark

To draw on the picture, we advise you to remove the mane from the drawing.

Reposition again the mane on the drawing, select it and then click on the **TypeArt** icon.

Select the mane and click on the **Section** icon.

Enter the section parameters of the mane and select the **Add** option in order to add the volume.



Masking

Principle

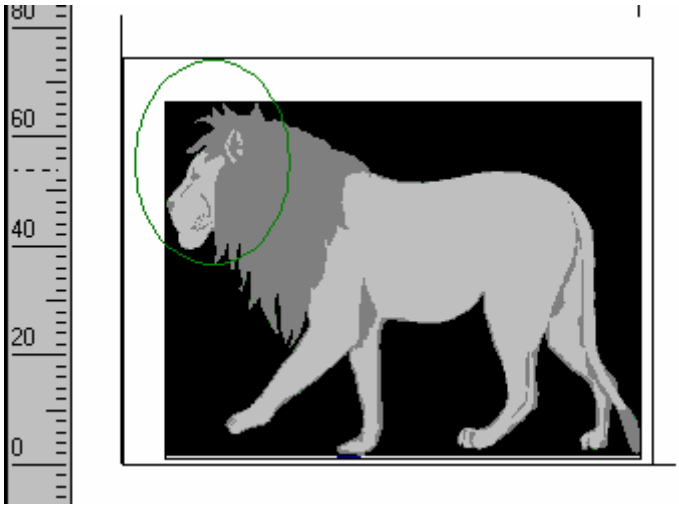
It consists in defining the limit of the working area with a tracing.

Example on a picture

or . Scan or import the picture with a **Bmp** or **Tiff** format.

Position the picture on the page, preferably in the lower left-hand corner and adjust the size as desired.

Trace the periphery of the area to be created in TypeArt with the **Sketching tool** or with the geometrical tools (circle, rectangle, etc).

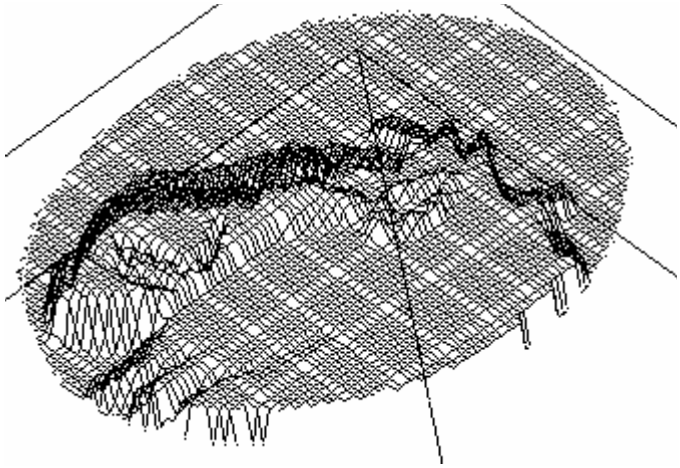


Run TypeArt and make sure you give the correct dimensions to the TypeArt object, so that the mask fits into the box.

Select the picture and then the mask. Next click on the **Grey level texture** icon.

Enter the desired elevation values for the gray levels.

The volume is applied only on the surface of the masked.



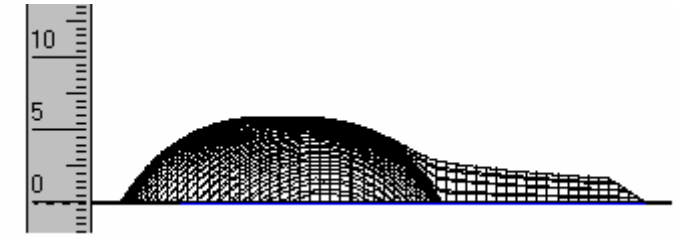
• *Canceling a section*

If the section is not satisfactory, you can undo the last three actions.

Click on this icon and select another section.

• *Choosing a raised or recessed section*

Section in relief: Male shape
The **TypeArt** object is automatically positioned at Z = 0 mm. There is no particular positioning of the engraving to be done.



Recessed section: Female shape
To create recessed objects, first make a section relief and then depending on your design, use the mirror function either along the X, Z axis or along the Z axis only.

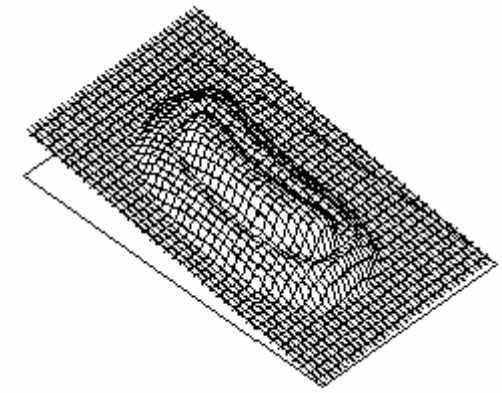
• *Reference plan of a TypeArt object*

Use this function to define the reference plan of a **TypeArt** object with the help of three markers. It is very interesting with digitized files because you can align horizontally a **TypeArt** object that couldn't have been digitized perfectly horizontal.

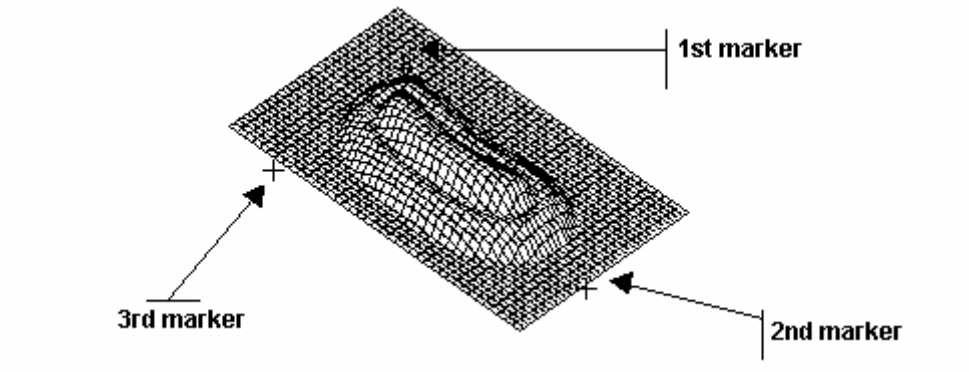
Once you've created your **TypeArt** object, switch back to the **CAD** module by clicking on the **Selection tool** icon . Place three markers on the TypeArt object and run the **TypeArt module** again by clicking on its icon . As soon as you are back in the **TypeArt module**, select the three markers (selection order isn't important).

Click now on the icon **Align plan 0** . The three markers are projected on the TypeArt object (every marker has its own height). The plan represented by these three markers is aligned horizontally. It became the reference plan.

Example



Before alignment



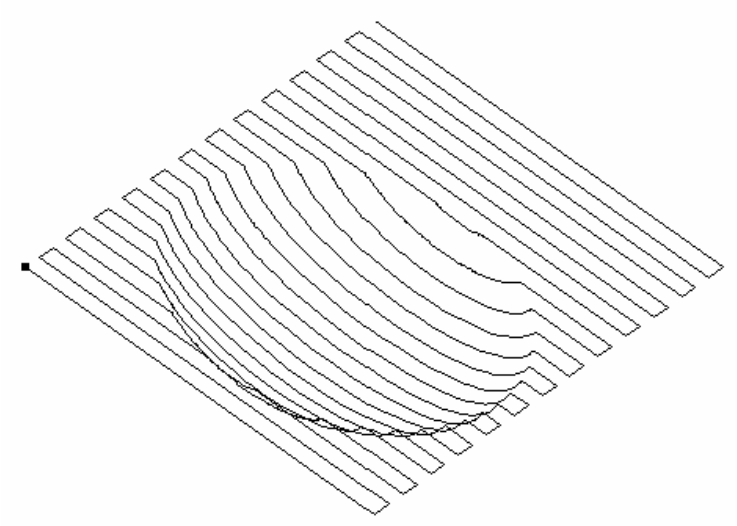
After alignment

- ***Canceling the machining of the TypeArt box (for recessed engravings)***

To avoid the machining of the creating a TypeArt object box, especially for recessed engravings, under the **Machining** menu, click on the **Cote** command.

YZ plane *3D engraving view*

Recessed object
External part

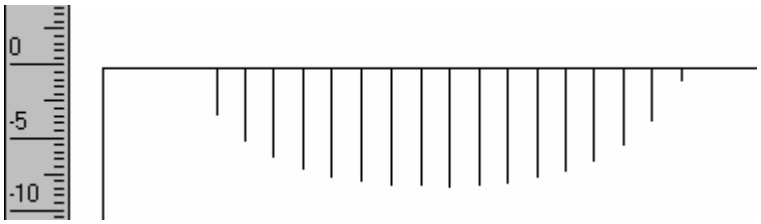


Visualization of the TypeArt tool path

Modifying the Z beginning of machining

In order that the tool path made by your machine will begin only inside the recessed part of your TypeArt, enter –0.01 mm in the **Z security start of machining** field found in the **Machining parameters** box that you can access through the icon **Parameters…** in the dialog box **Machining**.

Result sent to the machine:



YZ plane view of the machining path *3D ISO view of the machining path*

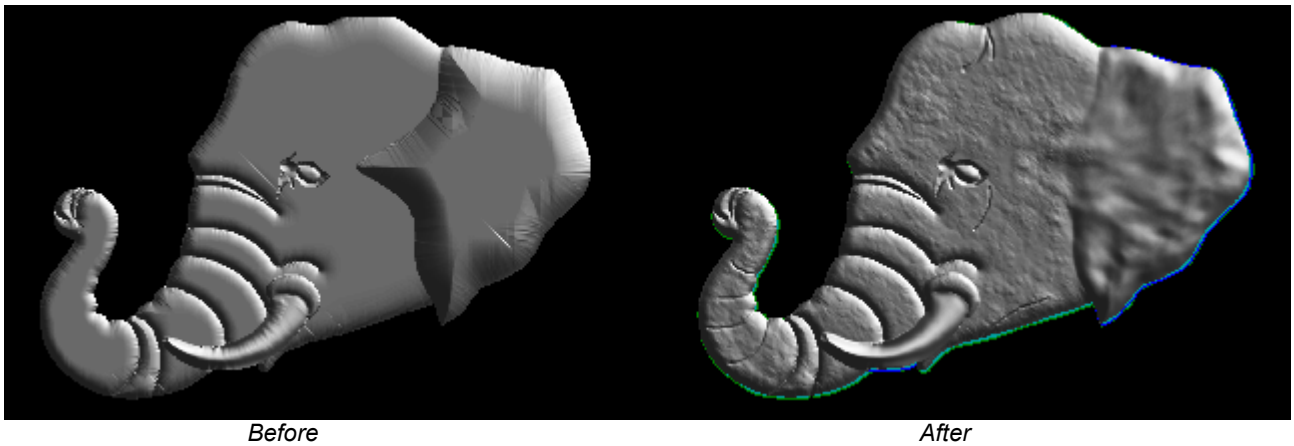
- ***SculptArt***

The **SculptArt** module is made of brushes that allow to add material, to push it, to attract or to twist it with the mouse.

Be careful: this option requires a powerful configuration (processor 350MHz, RAM 128 MB recommended)

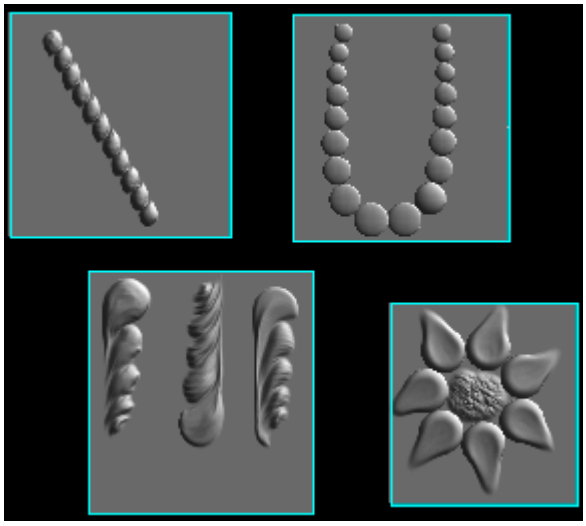
Examples:

Touched up TypeArt:



Before *After*

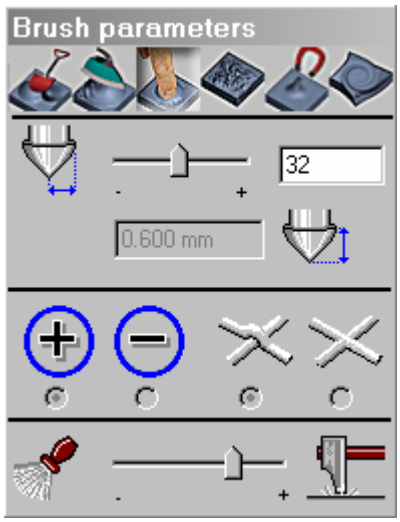
*Creation thanks to the **Brush** tool:*



You'll see that this tool can be used in an intuitive way and is totally dedicated to the artistic creation. Make many trials and have a lot of fun!

- **Use**

Select the TypeArt object and click on the **SculptArt** icon . Two windows appear on the screen: a visualization window and the following parameterization window:



- **Effects**

By clicking on these different icons, you will obtain the following effects:

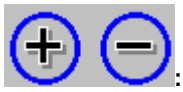
	Allows to add or subtract material according to the fact that **+** or **–** is selected (see further).
	Allows to smooth material.
	Allows to push material.
	Allows to add noise on a TypeArt object.
	Allows to attract material like a magnet.
	Allows to twist material (rotative movement).

- **Parameters**

All tools can be parameterized:

– **Tool radius** **:** Move the slider or enter a value between 2 and 64 (unit: pixels). This

radius is shown in the visualization window (can be applied to the 6 actions).

- **Profile height** **:** Enter the desired height of the profile (can be applied to the actions 1 and 4).
- **Recessed or raised**  **:** Tick the box corresponding to the effect you would like to produce: to add or subtract material (can be applied to the actions 1, 4 and 5).
- **Intersection mode** **:** While working, without releasing the mouse, choose between adding material and fusionning it on the intersection (can be applied to the actions 1, 2, 4 and 5).
- **Tool force:** Change the tool force by moving the cursor (can be applied to the actions 3, 5 and 6).
- **Tool shape:** Click on an icon to choose the shape of your tool (can be applied only to the actions 1 and 5).

- **Handling**

The **SculptArt** visualization window has a 2D and 3D view separated by a mobile bar.

To rotate the 3D view, click simultaneously on Ctrl and the left mouse button.

In 2D view, the mouse right-click enables to zoom in or out the image. Zoom buttons are also accessible at the bottom of the window.

The arrow appearing on the 3D view corresponds to the normal, i.e. to the orientation of the material in relation to the center point.

Close the window thanks the menu or the closing cross.

By combining all effects, you will obtain wonderful creations.

Transformation tools in TypeArt

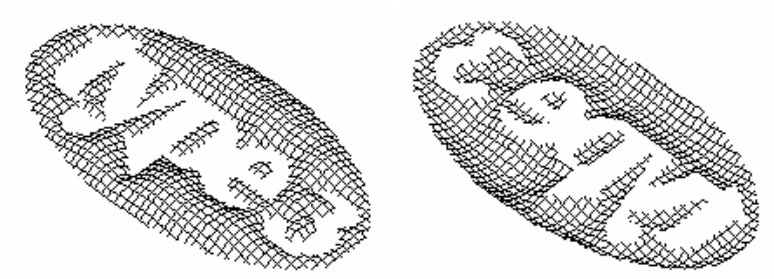
Running transformation tools in TypeArt

You can get access to the different transformation tools thanks to the **Transformation tools** icon in the main (left) toolbar of the **TypeArt module**. Click on this icon to display the **Transformation** toolbar (see each corresponding explanations in the following sections).

- ***Mirror***

Mirror on X, Z
This icon allows you to make directly a mirror image along the X and Z axis (for molds or recessed parts of embossing).

Example

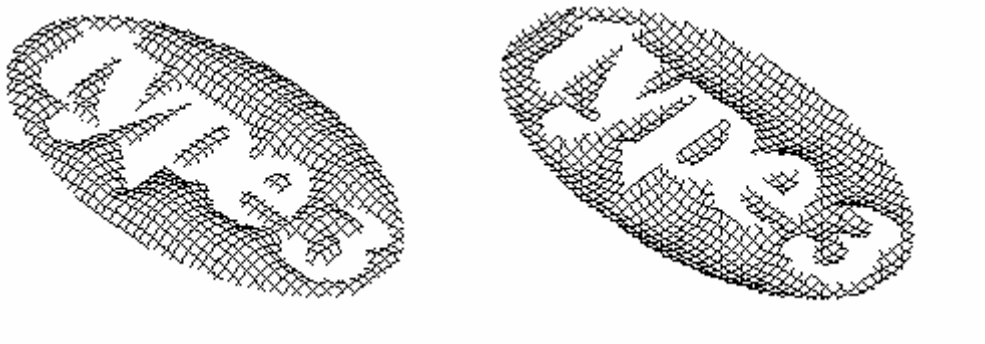


Relief *Recessed*

You can also do a 100% drop along the Z axis only.

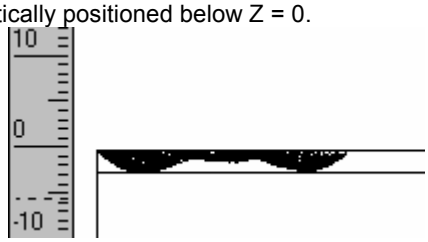
+ **Ctrl Mirror in Z**
The icon **Mirror**, used with the **Ctrl** key of the keyboard, allow you to create a mirror image along the Z axis. While pressing the **Ctrl** key, click on the icon.

Example



Raised *Recessed*

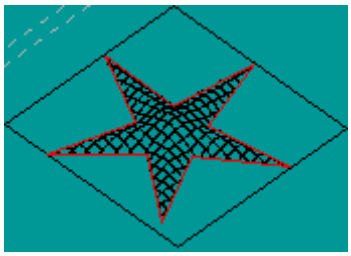
In **2D YZ view**, the object is automatically positioned below Z = 0.



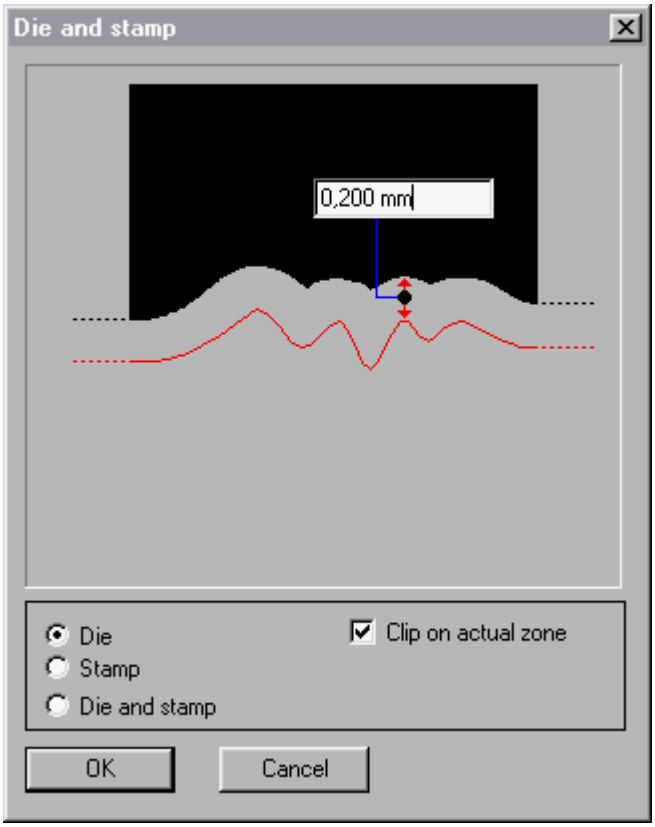
- ***Dies and stamps***

This new function allows to transform a **TypeArt** object into a die, a stamp or both (in that case, it consists in keeping the original object). Moreover you can insert an offset between the created objects in order to take into account the surplus material needed for stamping.

Select the **TypeArt** object.

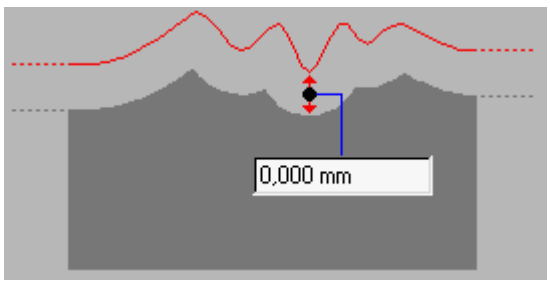


In the **TypeArt** toolbar, click on the **Die / Stamp** icon . The corresponding dialog box is displayed on the screen.



Select the desired option(s). The dialog box immediately takes the selected option into account.

*Example with **Stamp** as selected option*



Remark

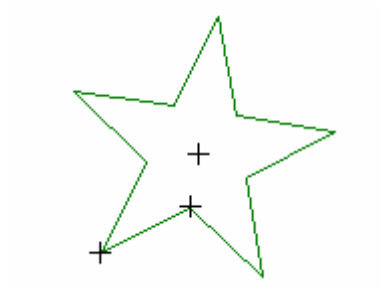
Enter the desired value in the numerical field to create an offset. Leave the value 0 if you don't want to create an offset.

- ***Plan defined by three points***

The interest of this function is to create inclined plans on a design with the help of three markers. Such plans are described as *Facets*.

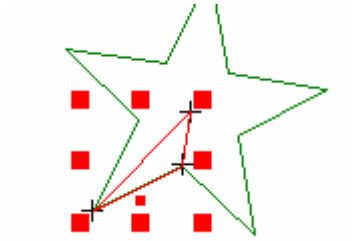
In the **CAD module**, place three markers that will define the limits of your inclined plan on the design.

Example



Choose then the Snap option **Marker**. Select the option **Snap mode…** in the **Edit** menu (or strike the **F3** key). Click then on the **Sketching tool** icon and draw the contour of the inclined plan (passing through the three markers). You can modify the height of one (or more…) marker to incline the plan. Select the new created plan and click on the icon to switch to the **Shaping mode**. Double-click then on each marker which height has to be modified. Enter the desired height in the **Z** field of the displayed dialog box **Point attribute** and validate by clicking on **OK**.

Example

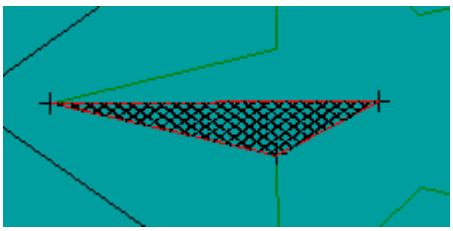


Switch back to the **TypeArt mode** by clicking on the corresponding icon . Back in this module, select the new modified curve (contour) and click on the **Facet** icon . The current **Combination mode** applies.

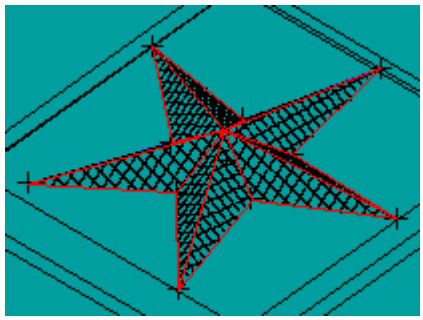
Remark

In the example shown hereafter, the same operation has been repeated as many times as there are vertices on the star.

Example



Operation executed once



Operation repeated

Duplication tools in TypeArt

- ***Running duplication tools in TypeArt***

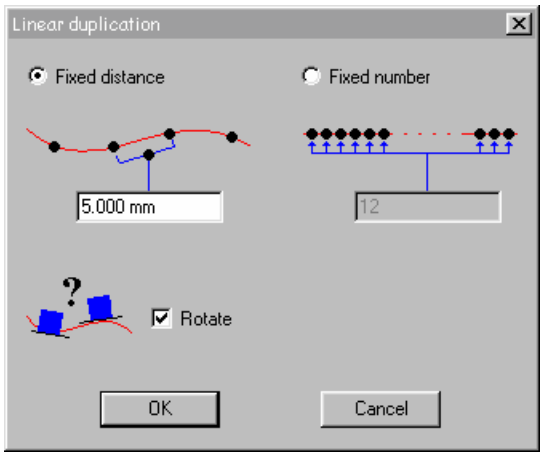
You can access to the different duplication and texturing tools thanks to the **Duplication** tools icon in the main (left) toolbar of the **TypeArt module**. Click on this icon to display the **Duplication** toolbar (see each corresponding explanation in the following sections).

- ***Pearling***

This function permits you to duplicate any kind of TypeArt object on a curve. It is especially useful for the creation of pearling, friezes or any decoration set along a defined contour.

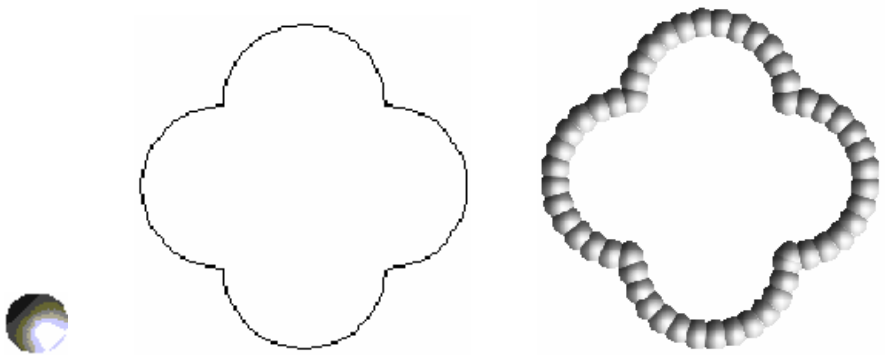
The principle is to create a first TypeArt object and then a second one around the curve on which the copy has to be done.

First select the TypeArt object to be duplicated and then the curve. Click then on the **Pearling** icon. The Combination mode dialog box is displayed on the screen allowing you to define the way the created TypeArt will be positioned according to the original one. Validate this intermediary choice by clicking on **OK**.
The **Linear duplication** dialog box is displayed on the screen.



In this dialog box, enter the desired number of copies or define the distance between each copy. Activate the **Rotate** option if you want to rotate the item.

Example



1st TypeArt *Duplication curve* *Final TypeArt*

Special effects in TypeArt

• *Running special effects in TypeArt*

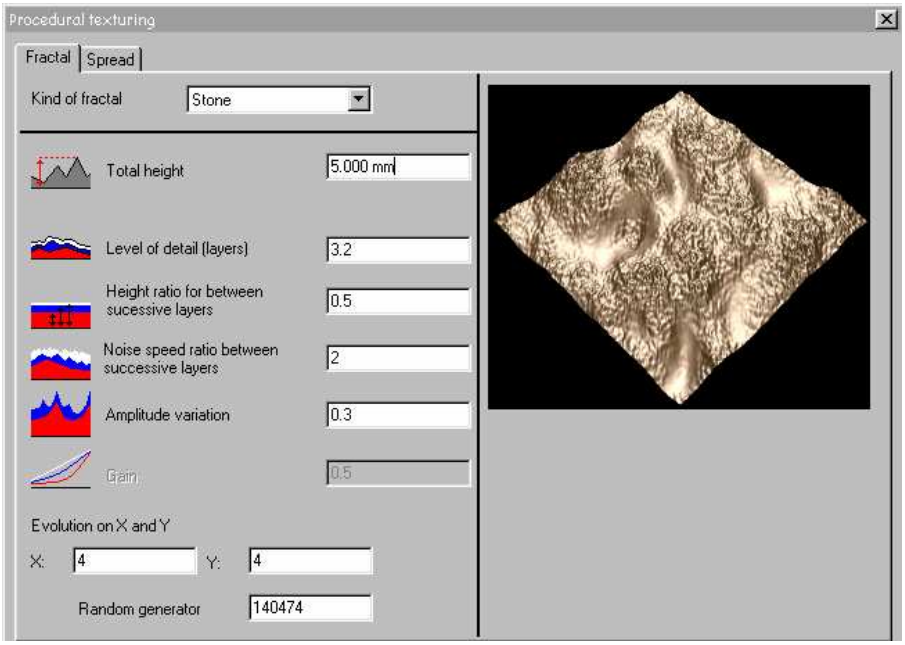
You can access to many special effects thanks to the **Special effects in TypeArt** icon in the main (left) toolbar of the **TypeArt module**. Click on this icon to display the **Special effects** toolbar (see each corresponding explanation in the following sections).

• *Procedural texturing*

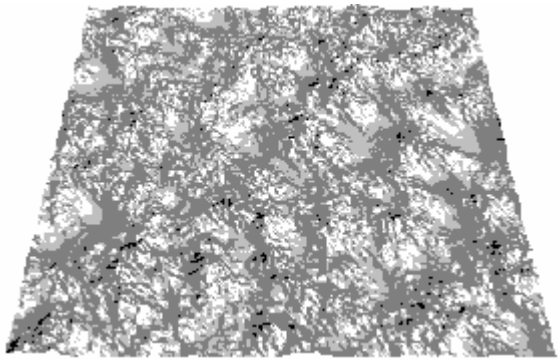
Make use of this feature to create lifestyle textures on a part or on the whole TypeArt object.

Click on the **Procedural texturing** icon . The current **Combination mode** applies. The **Procedural texturing** dialog box is displayed on the screen in which you can choose among two kinds of texturing:
Fractal
Spread

The fractal-type procedural texturing
The **Fractal-type** corresponds to a kind of depression of which the total height, level and repetitions of details, amplitude variation, etc. can be parameterized.
The evolutions of these depressions can also be parameterized on the X and Y axis. Moreover, you can assign a random factor (called **Random generator**) for the final result.

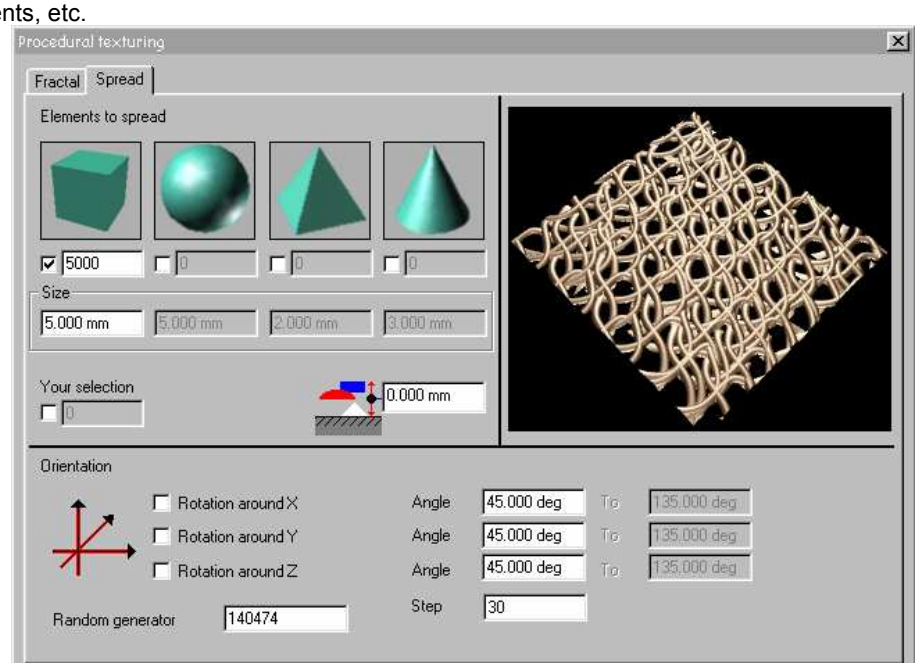


Example of a Fractal-type texture

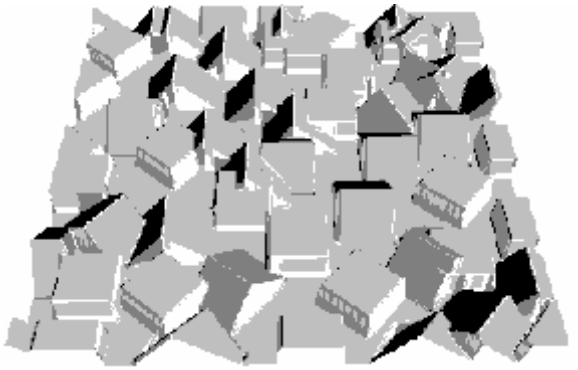


The spread-type procedural texturing

The **Spread-type** corresponds to the spreading of simple geometrical elements (box, sphere, pyramid, cone) or selection (i.e. any TypeArt object previously created) defining the number of copies, the rotation in X, Y or Z of copied elements, etc.



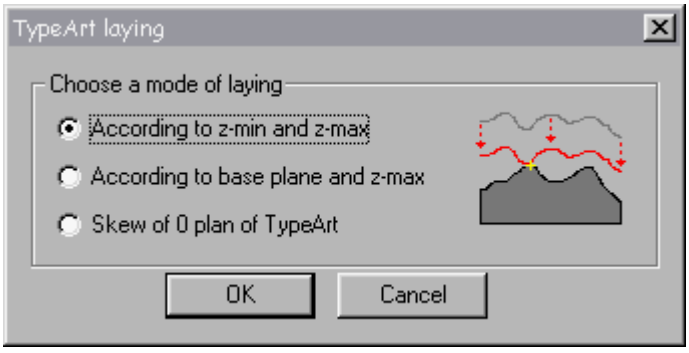
Example of a spread-type texture



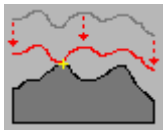
- ***Laying a TypeArt-object on another one***

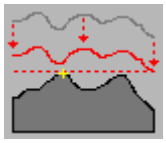
It is possible to lay a TypeArt-object on another one using three methods.

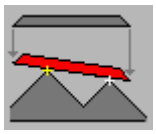
Select the TypeArt object and click on the **Lay** icon from the **Special effects** toolbar. The **TypeArt laying** dialog box appears on the screen.



Activate one of the three options. The dialog box picture changes automatically.

 Activate the **Z-min and Z-max** option so that the lowest point of the TypeArt object to be laid won't exceed the highest point of the other object.

 Activate the **Base plane and Z-max** option so that the base plane of the TypeArt object to be laid won't exceed the highest point of the other object.

 Activate the **Skew of 0 plane** option so that the base plane of the TypeArt object to be laid will fit the skew of the other object.

• *Warping grid*

This feature allows you to deform a TypeArt in a defined envelope. It is particularly useful to create a 3D decoration flat and then to deform it in 2D in order to use it for boards decorations, etc.

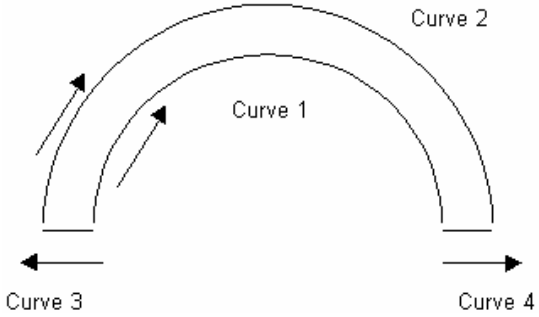
Principles

First create a TypeArt.

Example

Create then the envelope in which you want to create this TypeArt object. Obviously, you must strictly follow the different steps of the creation of an envelop (4 open curves). See the section **Creating an envelope** from the chapter **Special effects** about this theme.

Example



You want then to create a new TypeArt object using the first one and the created envelope. In the **TypeArt** module, select the TypeArt object to be deformed and select then the envelop respecting the order of selection.

Click on the **Warping grid** icon. The corresponding dialog box is displayed allowing you to define a cutting in X and Y of the TypeArt object to be deformed. For a smoothly-looking deformation, the cutting accuracy must be related to the TypeArt-object accuracy (for a 100 x 10 TypeArt, the cutting must be 100 x 10).
Once the calculation is done, the TypeArt object is deformed and is located in the envelope.

Result

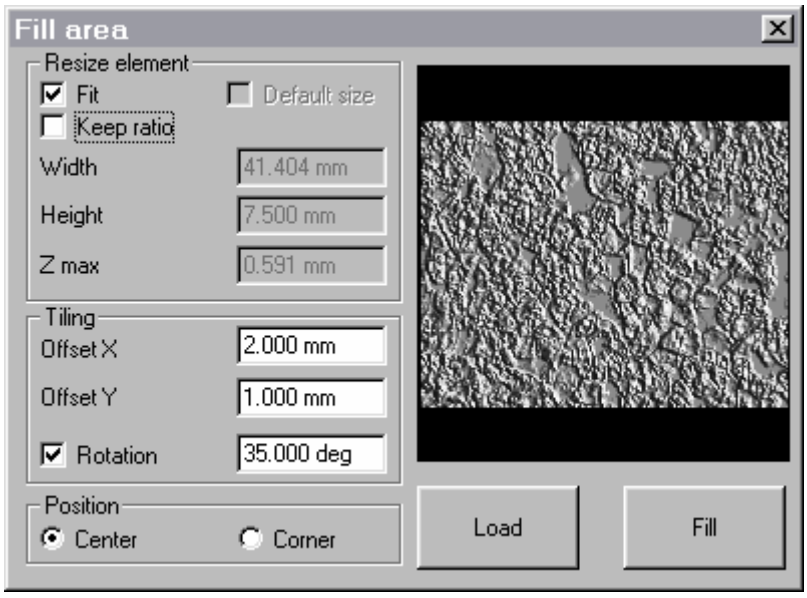
• *Fill with a texture*

This feature allows you to globally or partially fill a TypeArt object with a 3D texture. Just like before, it's particularly useful to create lifestyle surfaces, but this time from previously created 3D textures.

The texture database is made up of TypeArt objects of all kind. For instance, the objects could have been created by gray-level sampling and exported with the specific TypeArt format ***.vna**.
The software is delivered with many textures that can be enhanced with your textures.

Method

Select the contour to be filled with a texture (if you don't, the whole TypeArt object will be filled). Click now on the **Fill area with texture** icon . The corresponding dialog box appears on the screen.

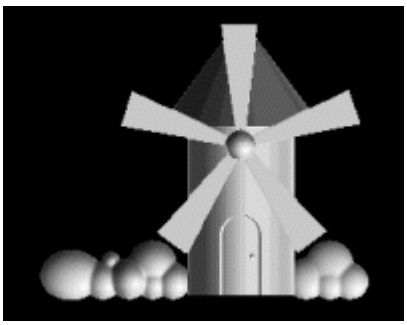


Load
Click on this button and select the texture with the ***.vna** format in the displayed dialog box.
Fill
Click on this button to apply the texture on the TypeArt object (partially or globally). The current **Combination mode** applies. Select the desired mode and validate by clicking on **OK**. The TypeArt object (or a part of it) is filled with the texture.

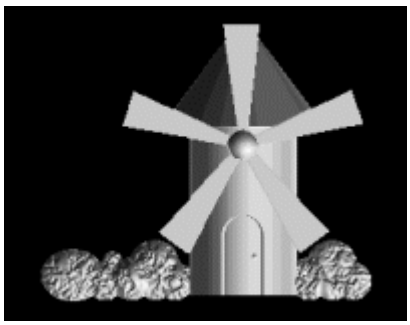
Parameters

Fit: activate this option to fit the size of the texture element to the dimensions of the selected contour.
Default size: activate this option to keep the dimensions of the original texture.
Keep ratio: activate this option to keep the ratio of the original dimensions in case you modify at less one of these dimensions (Height, width or Z: each of these parameters has got its associated numerical field to be filled).
Offset X / Offset Y: activate this option to define a spacing in **X** and **Y** between the copies of different texture-blocks.
Rotation: activate this option if you want to rotate the texture pattern, and enter the angle.
Center / corner: select one of these two options to define the Starting point of the duplication of the texture blocks on the TypeArt object, that is to say from the center or from the lower left corner).

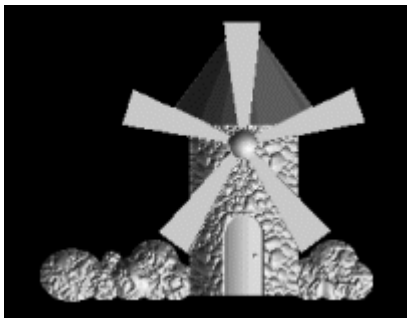
You can see below what you can get from an original TypeArt object.



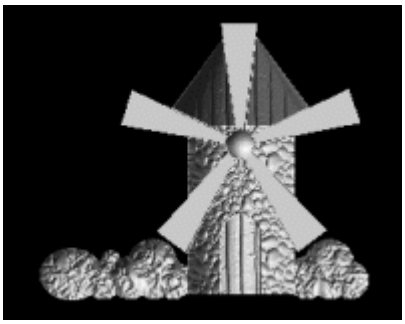
Original Object



First step



Second step



Third step

- ***Polishing***

To erase imperfections, you can effectuate polishings, especially on very rounded shapes.

Polishing tool
Click on this icon to activate the polishing function.
You can polish several times in a row or you can enter in the field a predefined number of passes.

The polishing is executed on the whole **TypeArt** object or only on a selected area.

Remark
Do not use this tool too much, otherwise you may loose some details!

- ***Gray-level texture***

This feature allows you to increase or decrease the contrast of a bitmap picture that has become a TypeArt object.

Click therefore on the **Gray-level texture** icon . The **PhotoArt** dialog box is displayed, allowing you to play with the sampling of gray levels to create a section automatically, adjusting the depth dynamic according to the gray levels that exist actually in the image.

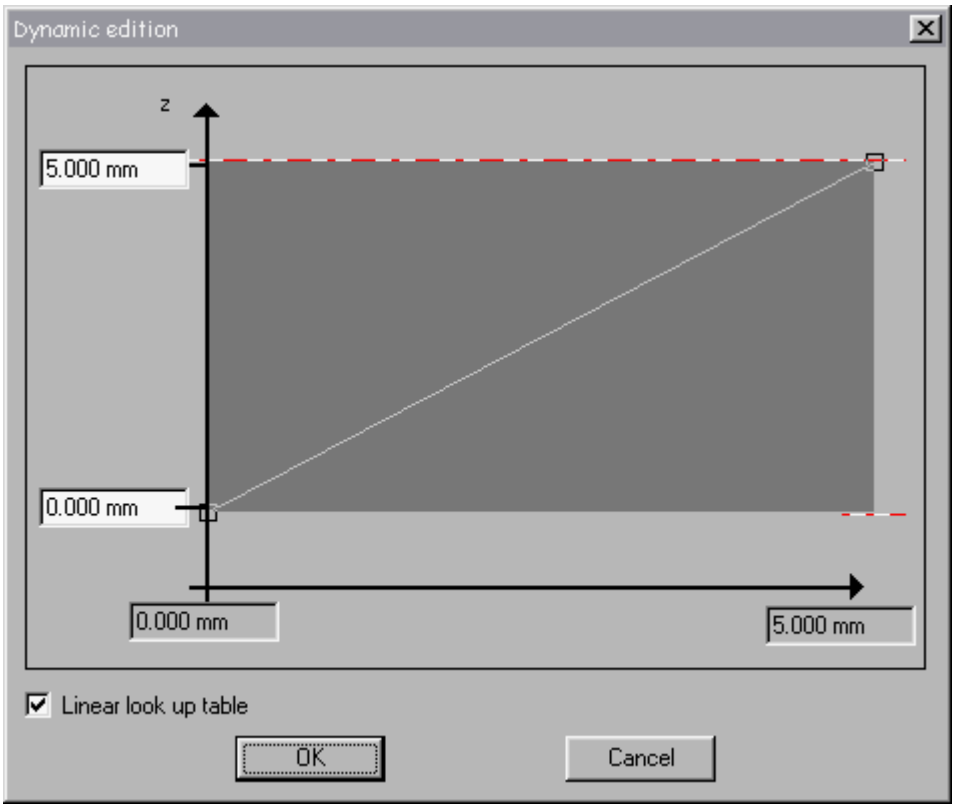
The sampling can be linear (that is to say the Z-value is constantly increasing), without contrast.
If the Linear look up table option isn't activated, a concave section smoothes the high areas meanwhile a convex section smoothes the low areas.

This function is often used to emphasize the contrast on a delimited portion of the image (mask). Select the image then the mask before clicking on (see Sections from a Bmp or Tiff file, above).

- ***Adjust dynamic***

This feature allows you to adjust the section of a TypeArt object, playing with Z Max and the section curving.

Click therefore on the **Adjust dynamic** icon . The **Dynamic edition** dialog box is displayed on the screen.



When the section is automatically created, Z Max can be unaccurate because of the size of the TypeArt and of the angle of the curve. The specified Z Max given with this function will be respected and the curving of the section adjusted.

The sampling can be linear (that is to say the Z-value is constantly increasing).
If the **Linear look up table** option isn't activated, a concave section smoothes the high areas meanwhile a convex section smoothes the low areas.

CAM tools for TypeArt

- ***Running CAM tools for TypeArt***

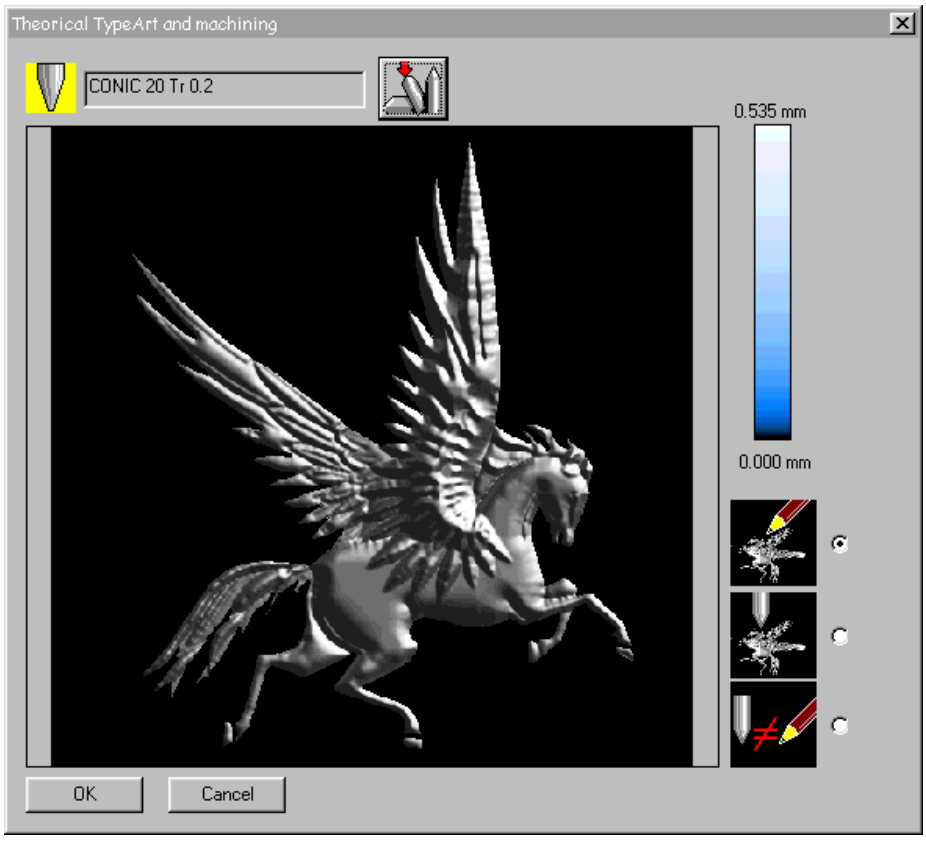
You can access to the different **tools for CAM** in TypeArt by clicking on the corresponding icon in the main (left) toolbar of the **TypeArt module**. Click on this icon to display the **CAM for TypeArt** toolbar (see each corresponding explanation in the following sections).

- ***Tool simulation on a TypeArt***

This feature allows you to simulate graphically the theoretical result of a TypeArt object after machining according to the tool choice. It's particularly useful to select the tool that fits the best to the object to be machined.

You access this function through the **Test for a tool** icon . Once you've clicked on the icon, the **Theoretical TypeArt and machining** is displayed on the screen.

Example



Click on the icon in the upper part of the dialog box to select a tool for the simulation.
In the displayed dialog box, select the desired tool and validate your choice by clicking on **OK**.
The vertical blue bar in the right part of the dialog box corresponds to the differences scale between theoretical object and machined object (cf. third displaying mode) that is to say the remaining material.
You must now activate one of the three displaying options (see below).

This is the default displaying option. It corresponds to the **theoretical model** (i.e. TypeArt object).

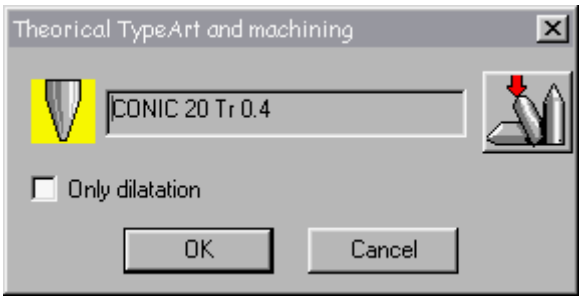
This displaying option corresponds to the **machined object**.

This displaying option corresponds to a kind of radiography of the differences between the theoretical model and the machined object. The display changes from dark to clear. Clear parts correspond to the ones with most of the remaining material. Dark parts correspond to the ones with less of the remaining material. Black parts correspond to the parts totally machined (in that case, there's no difference between theoretical and machined objects).

- ***Modifying a TypeArt object according to the tool***

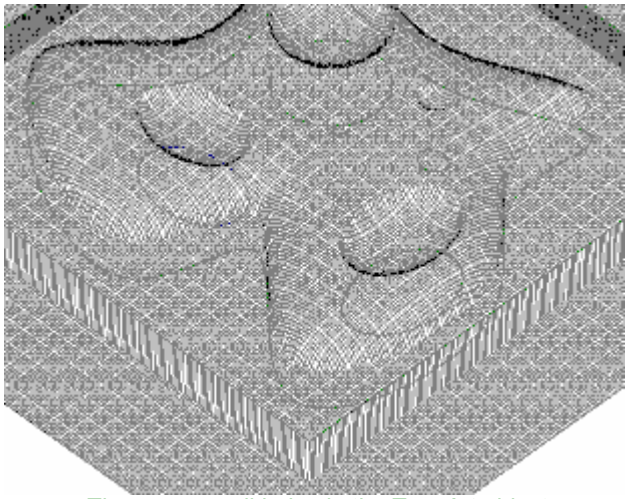
This feature permits you to modify a **TypeArt** object according to the choice of the tool.
In case the **TypeArt** object is made of vertical sections, these sections will be modified to take into account the cone-shape used for the tool. In case the **TypeArt** has some details that will be impossible to achieve because of the cone-shape of the tool and/or its half-angles, those details will be *filled in*.

Select the object and click on the **Modify for a tool** icon . The **Theoretical TypeArt and machining** dialog box is displayed.

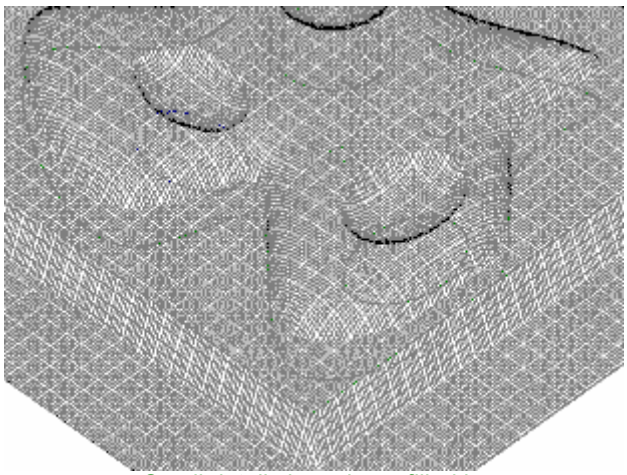


The selected tool in our example has the following properties.
Conical: 20°
Half-angle: 0.4 mm

Example



There are small holes in the TypeArt object

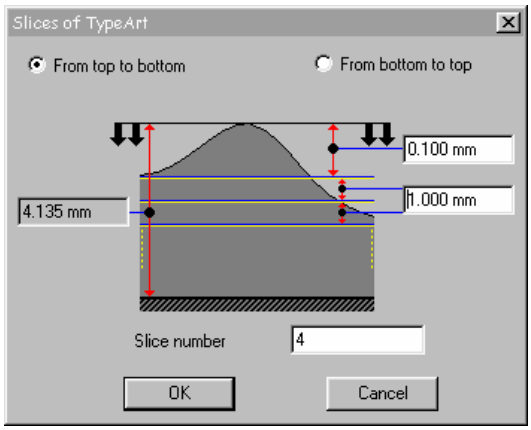


Small details have been filled in.
Vertical sections and tools now have the same angle

- ***Creating slices in a TypeArt object***

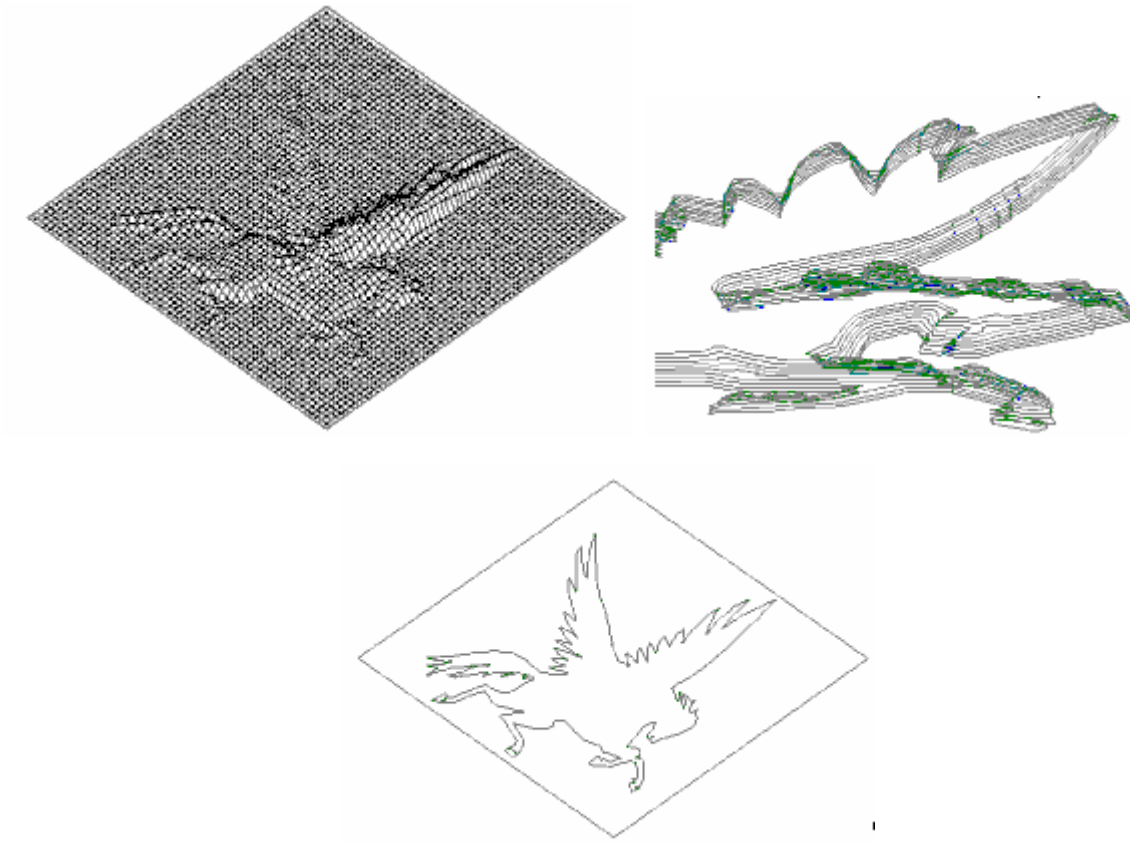
This feature allows you to create slices from a TypeArt object. It is particularly useful for getting back the contour of a digitized object.

Select the TypeArt object and click on the **Slices** icon . The **Slices of TypeArt** dialog box appears on the screen.



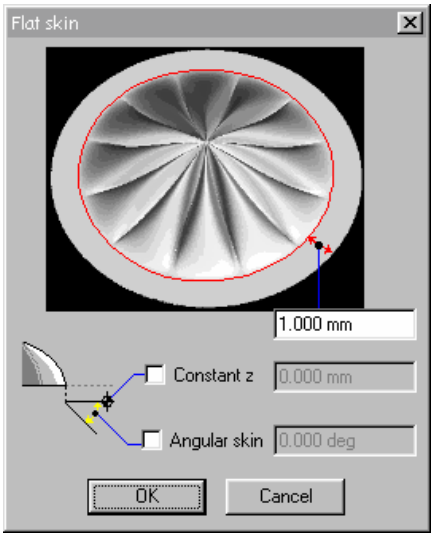
Choose among **From top to bottom** and **From bottom to top** to define the direction of the curves creation. The numerical field in the left part of the dialog box allows you to define the maximum height of the TypeArt object. The three numerical fields in the right part of the dialog box permit to define the distance between each slice, the Z start (from top to bottom) and the number of slices to be created (lower numerical field of the dialog box).

Example: if you ask for space by 0.01 mm, you obtain the contour of the TypeArt object



- ***Creating a flat skin around a TypeArt object***

This feature allows you to extend a TypeArt object. Select the TypeArt object around which you want to create a flat skin and click on the **Flat skin** icon . The corresponding dialog box appears on the screen.



Three different methods can be used to extend a TypeArt object. It can be extended:

- According to a defined angle; activate the **Angular skin** option clearing the field and allowing you to enter the angle value.
- In a flat way, at specific Z position; activate the **Constant Z** option clearing the field and allowing to define its position.
- In the continuity of the shape of the TypeArt object. Don't validate any of both previous options.

Visualization tools in TypeArt

- ***Running visualization tools in TypeArt***

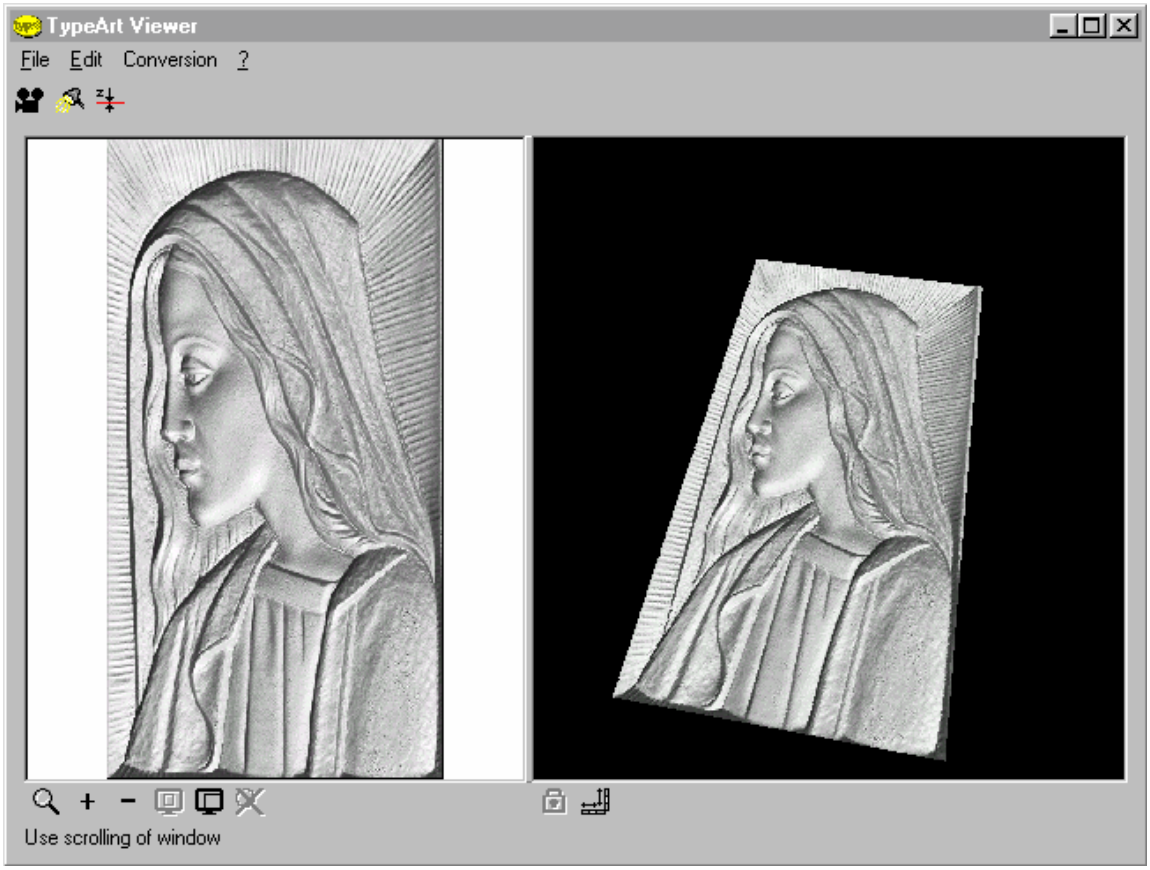
You can access the different visualization tools thanks to its dedicated icon in the main (left) toolbar of the **TypeArt module**. Click on this icon to display the **Rendering** toolbar (see each corresponding explanation in the following sections).

- ***The TypeArt Viewer***

TypeArt Viewer

This Rendering tool is very useful during a **TypeArt** creation. You can use this tool to have immediately a visual aspect of your creation.

Click on the **TypeArt Viewer** icon. The dialog box **TypeArt Viewer** is displayed.



Camera position

You can modify the position of the viewer by clicking on this icon. The camera symbol is displayed on the left screen of the **TypeArt Viewer** dialog box.

Example

Camera's position

Rendering

To modify the camera position, click with the left button of your mouse on the camera symbol (Blue square), and

while holding down the button, drag the camera. Release then, the left button of the mouse.

Example

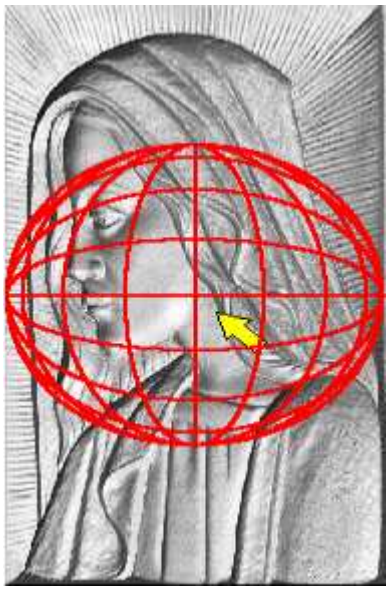
Camera position

Rendering

Light position

The **TypeArt Viewer** allows you to modify the light position. Click on the icon **Light position**. The light representation in the dialog box is red squared.

Example

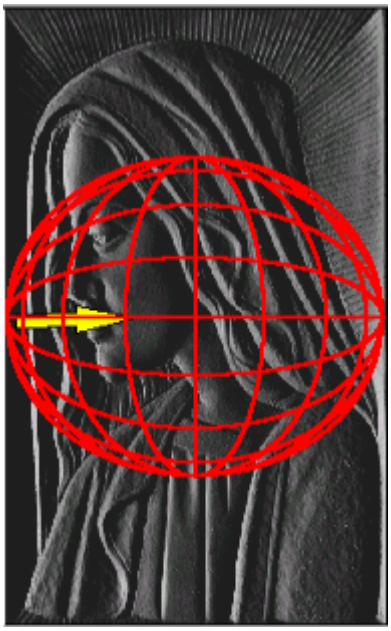


Light position

Rendering

Modify the light position with the cursor. Therefore, click with the left button of the mouse on the yellow arrow and while holding it down, drag the light until its desired position.
When the new position is reached, release the left button of your mouse.

Example



Light position

Rendering

Altitude of a point

Using this function, you can know the altitude of each point of your composition without having to quit the viewer. To enable the function, click on the **Altitude of a point** icon. The given altitude is the one of the point located under your cursor. The **Z** altitude is displayed in the status bar at the left bottom of the **TypeArt Viewer** dialog box.

Example

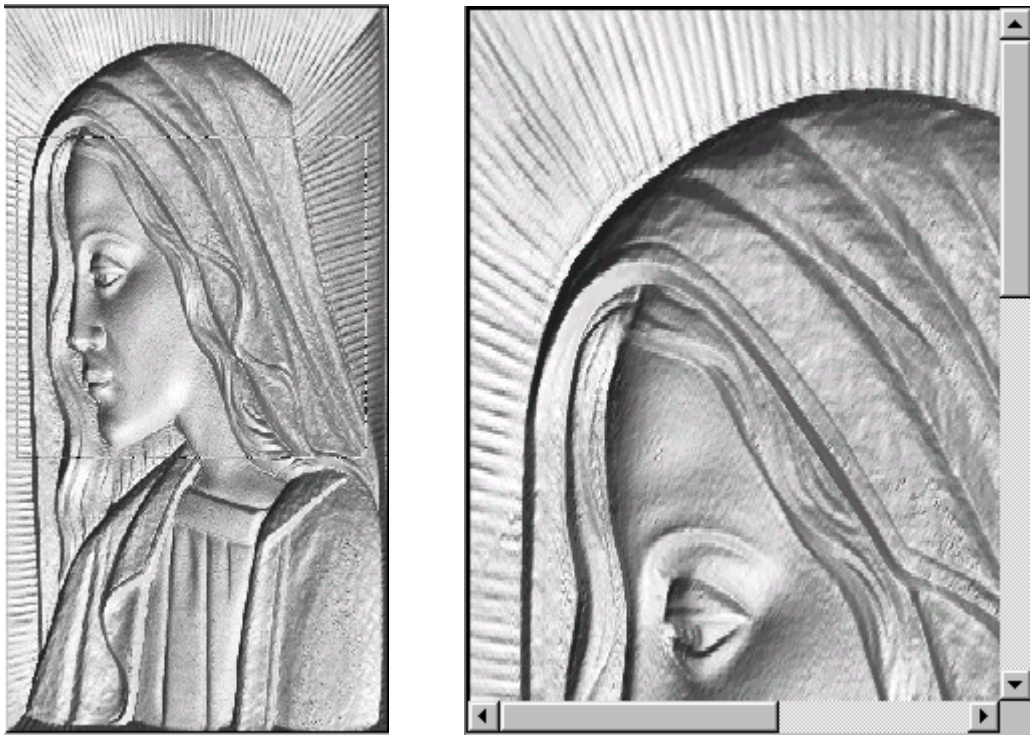
Display options

The left window of the **TypeArt Viewer** allows several display modes.

Surface zoom

This zoom tool allows to enlarge partially the picture displayed on the left window of the **TypeArt Viewer**. Click on the **Surface zoom** icon and drag the cursor of your mouse to define the part to be enlarged.

Example



Selection *Result*

To go back to the usual display mode, click on the **No zoom** icon .

Double zoom
Click on this icon to enlarge two times the size of the picture.
To go back to the usual display mode, click on the **Half zoom** icon .

Full page view
Click on this icon to display the picture in a full page mode (in that case, the full page is the left window of the **TypeArt Viewer**.

To go back to the original display mode, click on the **Scalable view** icon .

Modifying the size of a picture by free hand
To modify by free hand the size of a picture displayed in the right window of the **TypeArt Viewer**, click on the right button of your mouse. While holding it down, drag your cursor forwards to reduce the size of the picture or backwards to enlarge it.

Example

Rotating the picture by free hand
To make a picture rotate by free hand in the right window of the **TypeArt Viewer**, click on the left button of the mouse and while holding it down, drag your cursor. When the desired view is obtained, release the left button of the mouse.

• *Color map*

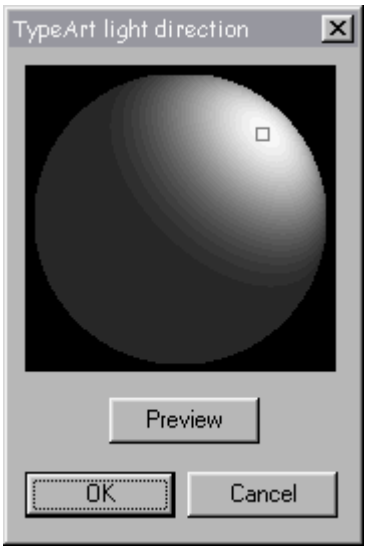
This feature allows to create a bitmap from the selected elements of a drawing. This bitmap will then be used to color different parts of the drawing so that you will obtain a color realistic rendering.

In the **TypeArt module**, select the different parts of the drawing and click on the **Color ma**p icon . A new bitmap is created next to the original drawing.

• *Light direction*

This feature allows you to modify the direction of light for the realistic rendering of a TypeArt object.

Click on the **Light direction** icon . The dialog box **TypeArt light direction** appears on the screen.



Using the mouse left button, click on the square, keep the button pushed and drag your cursor until you reach the desired position. Check the obtained result by clicking on **Preview**. Switch back to the original lightening by clicking on **Cancel**.

TypeView: running the module

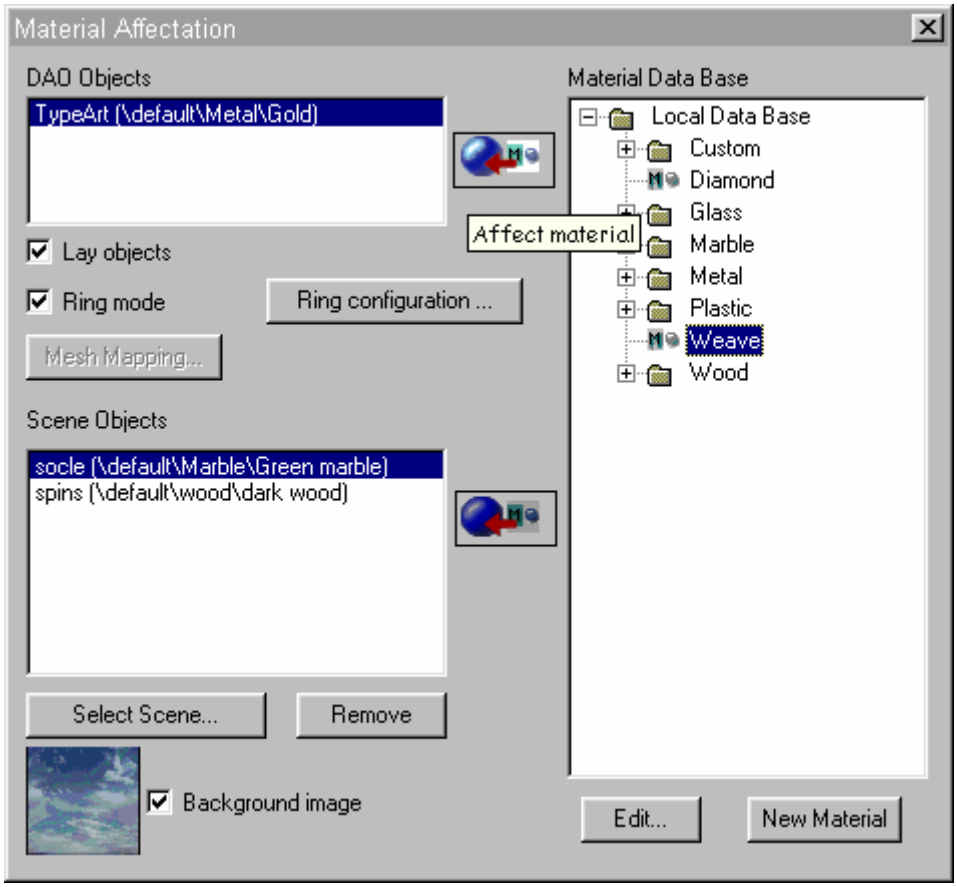
This feature is the realistic rendering allowing to obtain very high-quality pictures with:

- Various colors
- Shadow-effects, reflects…
- Transparency…

Select the object to be rendered and click on the **TypeView** icon from the TypeArt main toolbar. The **TypeView** dialog box is displayed on the screen. Just click on each symbol of the upper toolbar of this dialog box to access specific features of this rendering tool.

Material assignment in TypeView

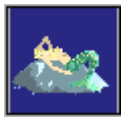
You can choose the kind of material you will assign to the rendered **TypeArt** object. Click therefore on the **Material assignment** icon in the upper part of the **TypeView** dialog box. The corresponding dialog box is displayed on the screen.

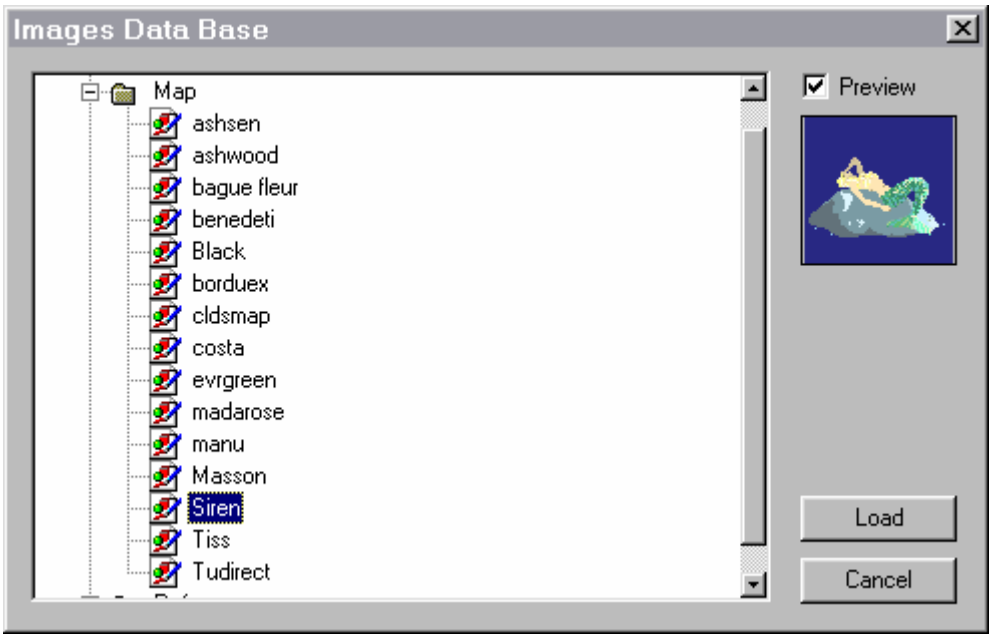


- To assign a material to a TypeArt object, select the TypeArt in the CAD Objects window, then select the material in the database on the right and finally click on the icon. By double-clicking on a material in the database window, or by the **Edit** button, the Material editor is displayed on the screen. It is possible to create a new material as well.

- The **Ring mode** checkbox allows you to enter parameters for rendering rings and bracelets.

- **Select scene**… allows to choose a background environment. Click on the **Select scene** button and double-click on **Socle** (or click once on **Socle** and then on **Load**). Choose an environment and click on the icon **Rendering!** to have an idea of the result.

- **Background image**…gives you the possibility to choose a background. Double-click on  after having selected the **Background image** in order to open the **Material Data base**. You can select an image or load your own bitmaps.



Click on the icon **Rendering!** to have an idea of the result. You can combine easily the **Select scene** and **Background image** options.

Material Editor

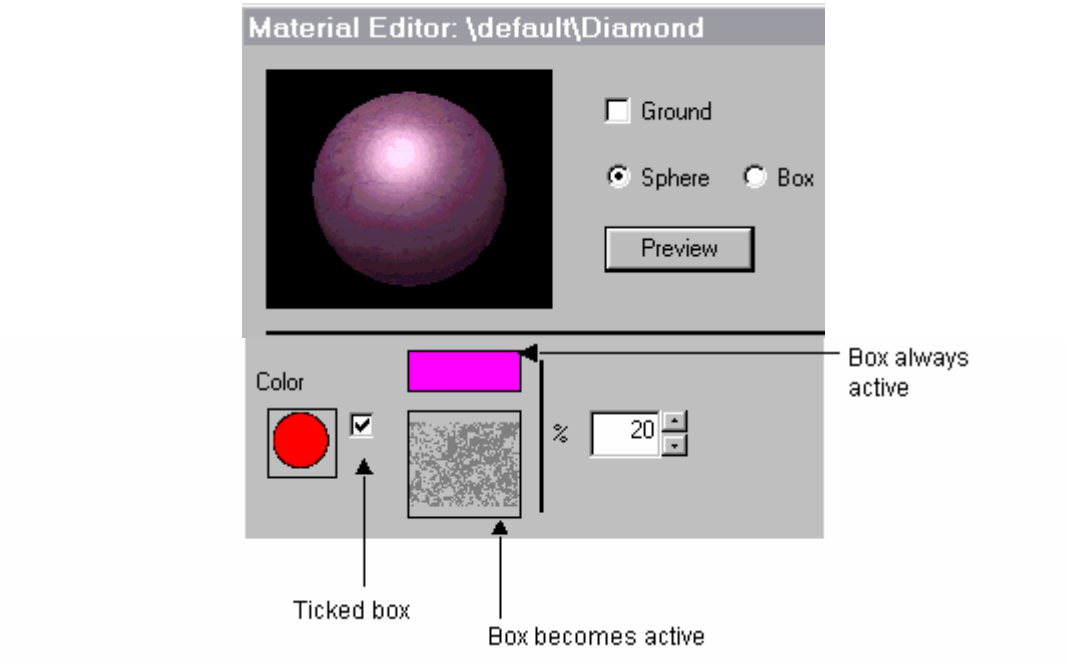
This dialog box gives you the possibility to combine many effects to infinity.

- **Color:**

Box not ticked (even rendering)	Box ticked off (uneven rendering)
Click on the box in order to choose the color you want to assign to the object. Push the **Preview** button to see the resulting effect.	Click on the box, which is now active. This allows you to open the database of images. Choose a pattern and click on **Load**. You can have an idea of the rendering with the **Preview** button.

You can combine both of the effects by assigning a **percentage** to each of them. This percentage gives the color in proportion to the used material.

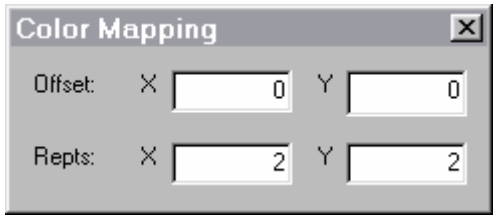
Example:



In our example, there is 80% of color and 20% of material.

NOTE:

Once the box ticked, it is possible to click on the boxes corresponding to the different effects (for instance: **) so as to open the dialog box allowing you to define the** mapping **or tiling: offset and number of repetitions of the object (value must be between –1 and 1). It holds for all effects except for the** Metal shininess**.**



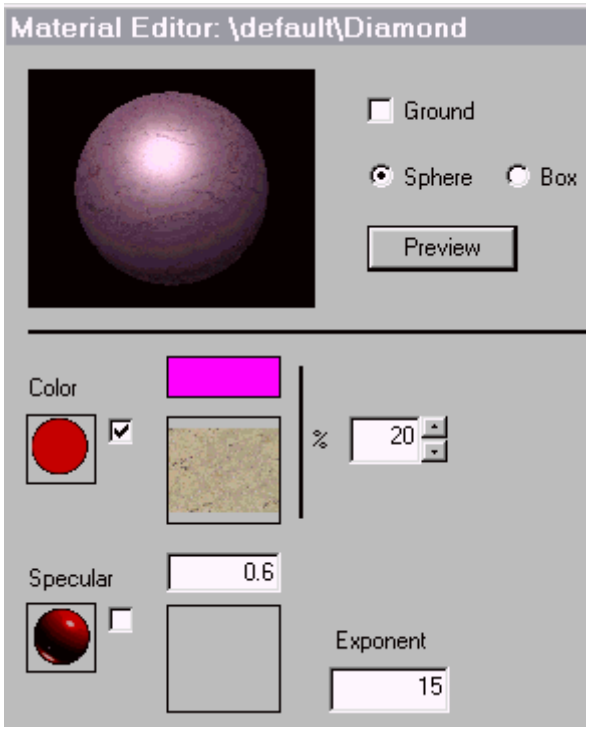
- **Specular:**

This option depends on the material property and allows you to parameterize the intensity of the luminous spot that appears on the object. The higher the value is, the bigger the spot gets.

Box not ticked (even rendering)	Box ticked off (uneven rendering)
Enter a value between **0** and **1**. Push the **Preview** button to see the resulting effect.	Click on the box, which is now active. This allows you to open the database of images. Choose a pattern and click on **Load**. You can have an idea of the rendering with the **Preview** button.

Both of the **color** and **specular** effects can be combined.
The exponent (value between **1** and **10,000**) allows to parameterize the size of the luminous spot. The higher the value is, the bigger the opening and the lower the light intensity gets.

Example:



- **Transparency**

A transparency effect can be assigned by using this function.

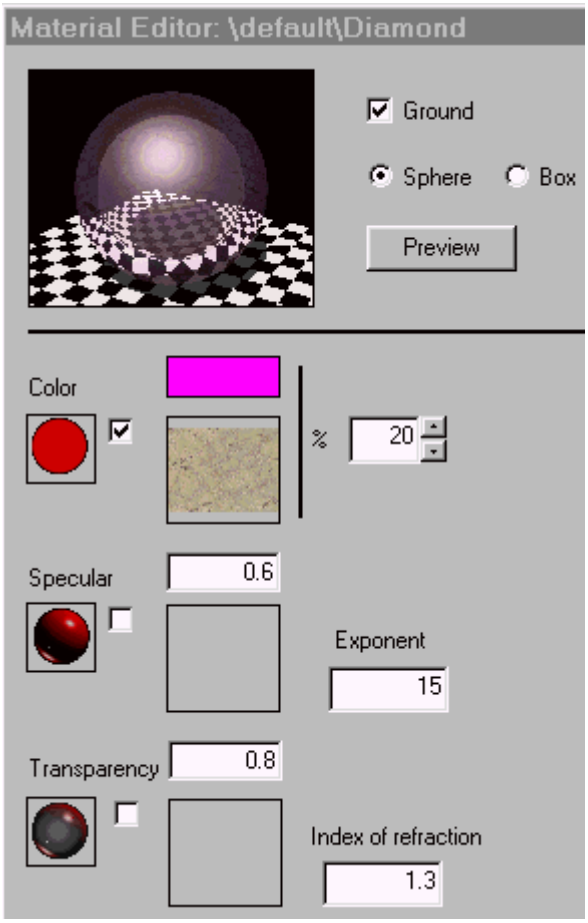
Box not ticked (even rendering)	Box ticked off (uneven rendering)
Enter a value between **0** and **1** (0 for an opaque object, 1 for a total transparency). Push the **Preview** button to see the resulting effect.	Click on the box, which is now active. This allows you to open the database of images. Choose a pattern and click on **Load**. You can have an idea of the rendering with the **Preview** button.

It is possible to vary the **index of refraction** of the material. The latter corresponds to the deformation of the image through a transparent object. Enter a value higher than **1**.

NOTES:

1. **The resulting effect depends on the original material.**
2. **It is advised to tick the box** Ground **to notice the transparency.**

Example:

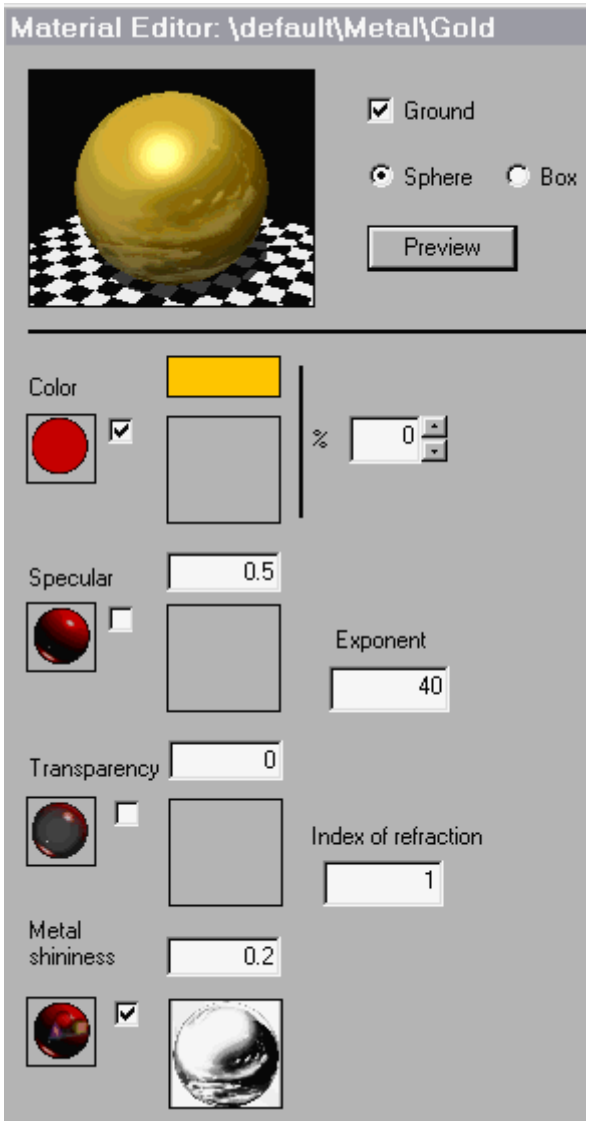


The index of refraction of our example corresponds to the one of water.

- **Metal shininess:**

This option can only be applied to metal effects. The box is automatically ticked off. The index changes from **0** to **1** (0 corresponds to a weak brilliance, 1 to the maximum).
Click on the box to open the database of images. In the **Refmap** menu, you can choose between 2 types of shininess. Confirm your choice by pushing the **Load** button. Use the **Preview** button to have an idea of the rendering.

Example:

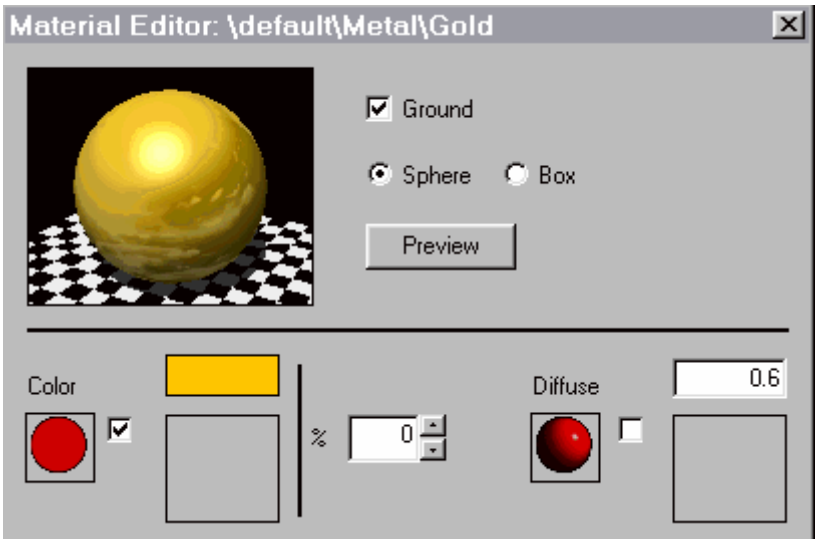


- **Diffuse light**

This function uses the property of the reflection of light by the material. As a result, you will obtain a luminous and diffuse field.

Box not ticked (even rendering)	Box ticked off (uneven rendering)
Enter a value between **0** and **1** (0 for low diffusion, 1 for a high one). Push the **Preview** button to see the resulting effect.	Click on the box, which is now active. This allows you to open the database of images. Choose a pattern and click on **Load**. You can have an idea of the rendering with the **Preview** button.

Example:



- **Reflection**

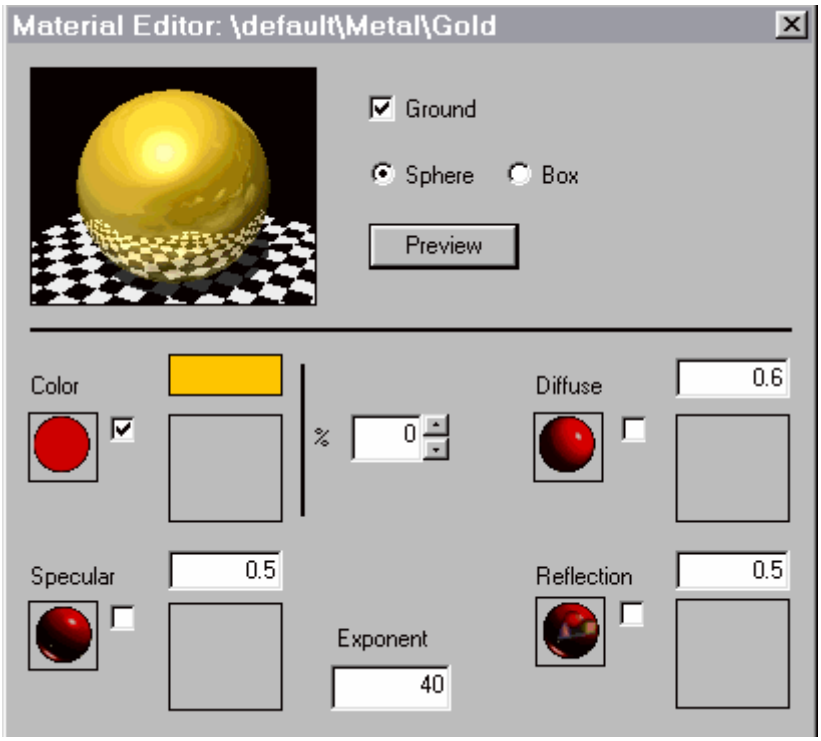
The reflection index of the material can be parameterized thanks to this function.

Box not ticked (even rendering)	Box ticked off (uneven rendering)
Enter a value between **0** and **1** (0 for low reflection, 1 for a maximal one). Push the **Preview** button to see the resulting effect.	Click on the box, which is now active. This allows you to open the database of images. Choose a pattern and click on **Load**. You can have an idea of the rendering with the **Preview** button.

NOTES:

1. **The resulting effect depends on the original material.**
2. **It is advised to tick the box** Ground **to notice the reflection effect. The environment is reflected on the object.**

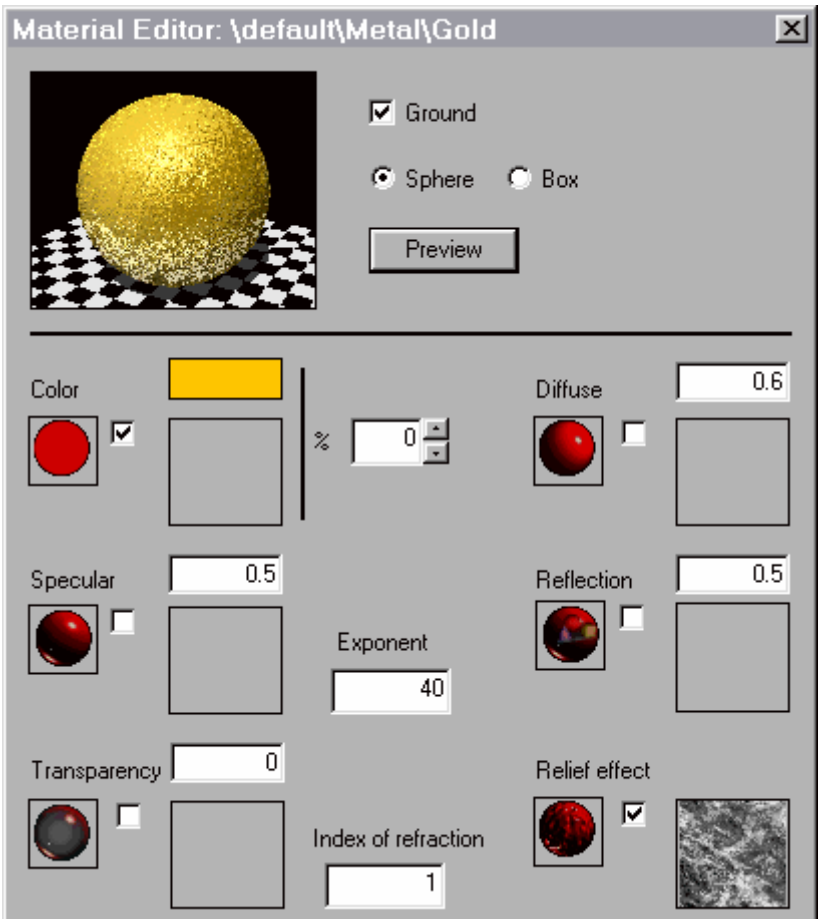
Example:



- **Relief effect:**

Tick the box to assign a relief effect to your object. Then click on the box which is now active to access the databank of images which allows you to choose between many relieves.

Example:



For this example, the "Crépi4" has been chosen.

This chapter does not pretend to be exhaustive as the combination possibilities seem to be infinite! We recommend you to make many trials. The unique limits this module have are the ones of your fantasy!!!

Create a material from a bitmap image

It is possible to complete the existing database of images with your own **bmp** (it is recommended to save the images in the directory **\Vision\Rtrace\Images\Map**).

The **TypeArt material,** which is the colored drawing of the TypeArt, can be imported. You can use all effects of transparency, brilliance, relief, color, etc. You have to import the image and send it to the TypeArt module. Use the tool to convert the image into a color bitmap. Select the colored image (right to the original bitmap) and send it to TypeView. Open the dialog box **Material affectation**. The TypeArt material appears in the **Material data base**.

To work and use this material, select the TypeArt Material with the right mouse button. While holding it down, drag it to the **Custom** directory. Release the left button of the mouse. The contextual menu is displayed on the screen. Choose the command **Copy** in order to copy the material in Custom. You can rename your image. By default, the image is called **New import**.

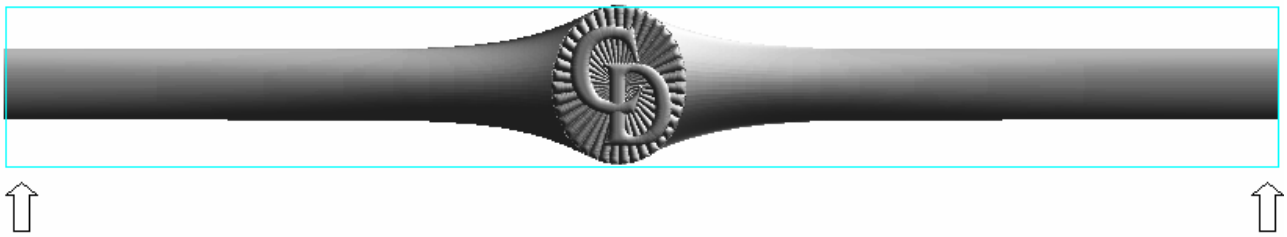
Ring mode in TypeView

The Ring function allows to produce realistic renderings of rings.
The principle is easy: you only have to make a TypeArt object, to send it in the TypeView module that will map it following the Z axis.

To realize a ring, the TypeArt object has to be made following some rules:

- It must be a flat object,
- The profile has to be edge to edge to the bounding frame if the mapping must be complete.

Example:



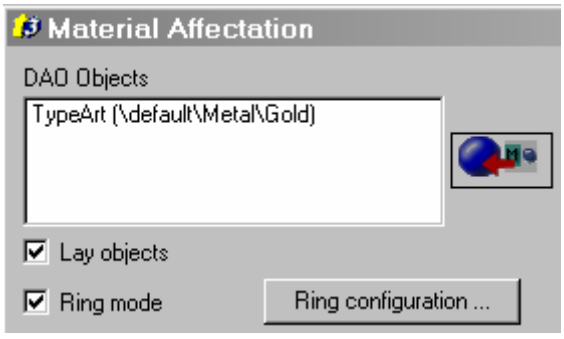
Flat TypeArt

Ring rendering

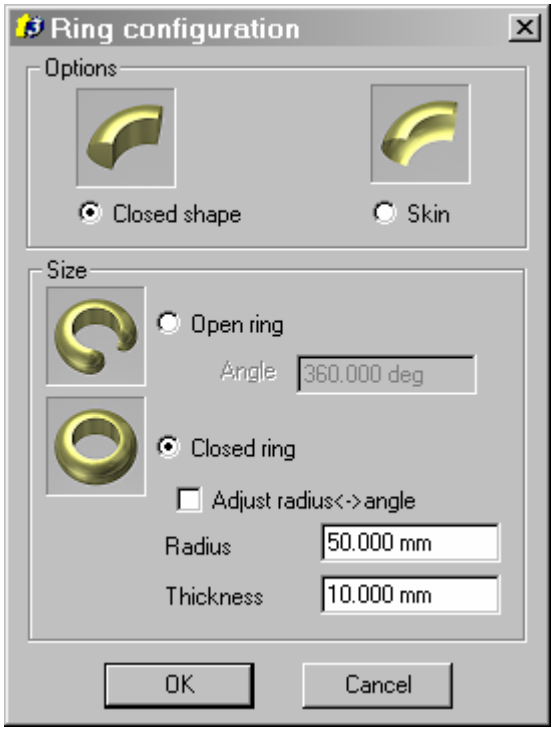
• *Launching of the Ring function*

This function is available in the dialog box **Material affectation** in TypeView.

To access to this function, tick the **Ring mode** box and click on **Ring configuration** to modify the mapping parameters.



The following dialog box is displayed on the screen:



- ***Adjustments of parameters***

  - **Options**

– Closed shape: the inner aspect of the ring is smooth,
– Skin: the inner aspect of the ring is hollow and has the relief of the TypeArt object.

  - **Size**

You can choose between a closed and an opened ring.

In case of an opened ring, you may enter the value of the opening angle.

In case of a closed ring, you can:
– Automatically adjust the radius in relation to the size of the TypeArt object,
– Enter the value of the radius,
– Parameterize the ring thickness (for a low thickness, enter 0.1 mm).

Note:
The numerous rendering tools of TypeView can be associated with the **ring** function, in particular selecting a color bitmap or adding a scene.

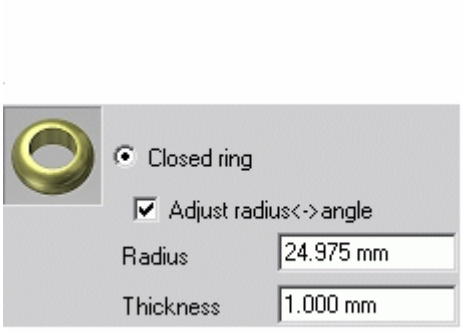
Example:

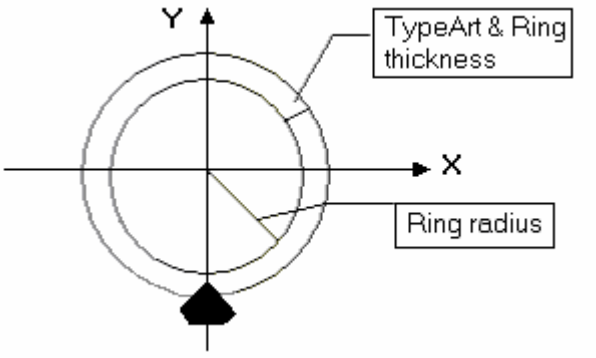
Rendering of a ring associated with a scene

It is also possible to use a surface in TypeView to associate it with your TypeArt object. In this case, the surface must be properly positioned. When the object has several facets (for instance a diamond), the latter must be grouped (**Ctrl** + icon **Combine**).

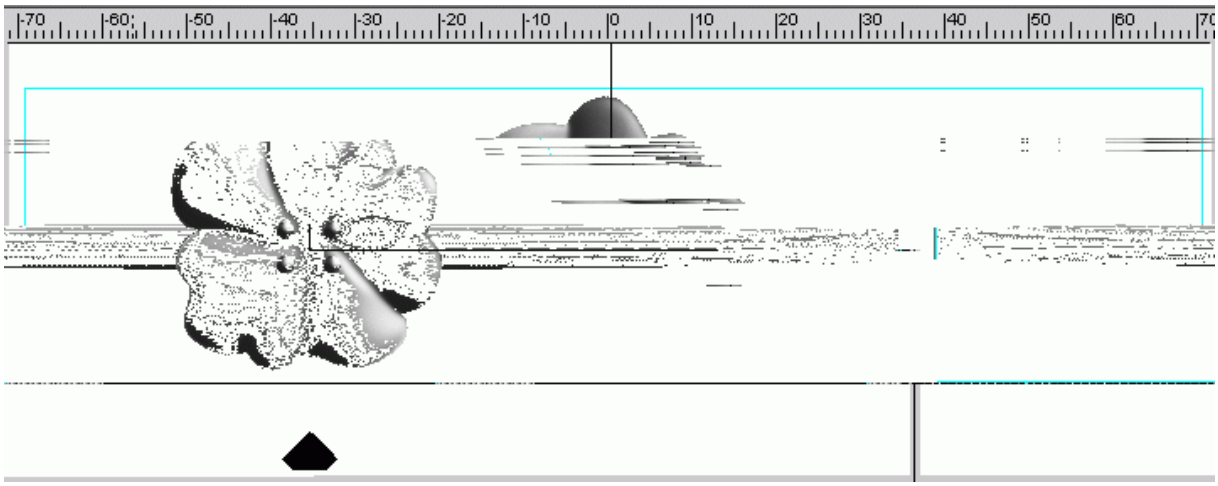
Example of a ring set with diamond

Place and position the surface in relation to the position of the TypeArt object with the tool **Flip plan** . Send the TypeArt object in TypeView and tick the box **Adjust radius <-> Angle** in the dialog box **Ring configuration**. The entered value corresponds to the radius of the cylinder and the TypeArt thickness.





The diamond is positioned on the Y-axis. To calculate its position properly, consider the value of the radius + the thickness of the TypeArt object, i.e. 25.975mm in our example. Adjust the position of the diamond by moving it slightly following the Y direction.



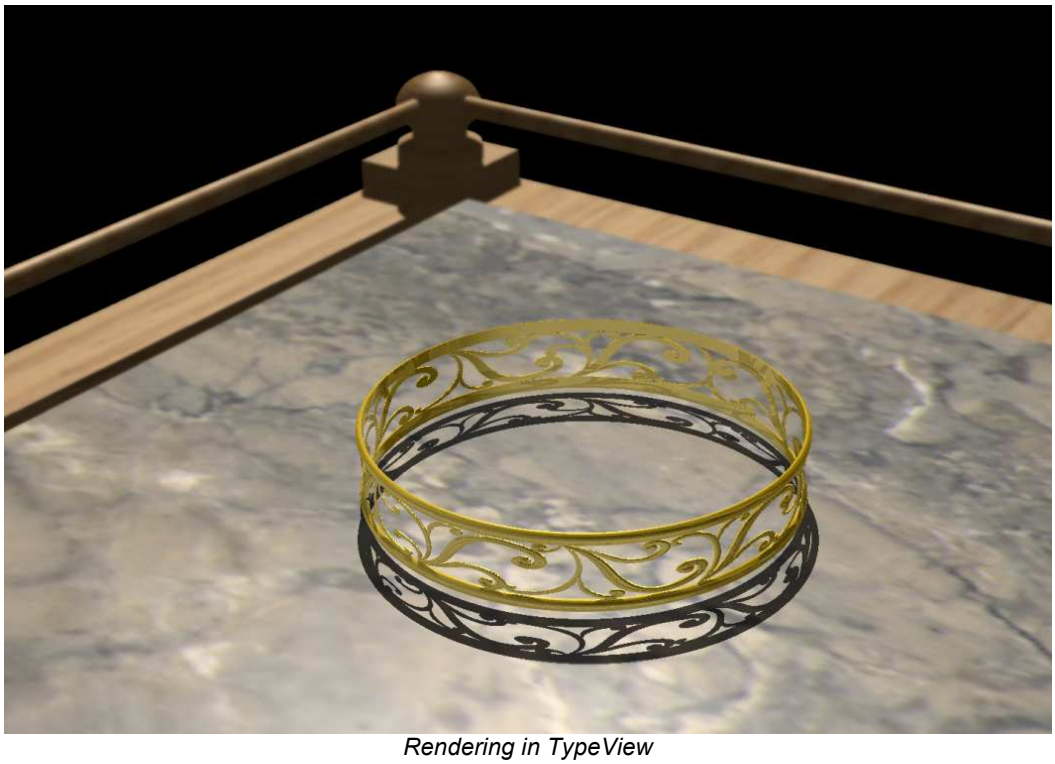
Positioning of the diamond before mapping

Rendering of the ring set with diamond

In case of a TypeArt with an openwork design, the rendering will take into account the aspect and all details you created.

Example of a bracelet:

TypeArt

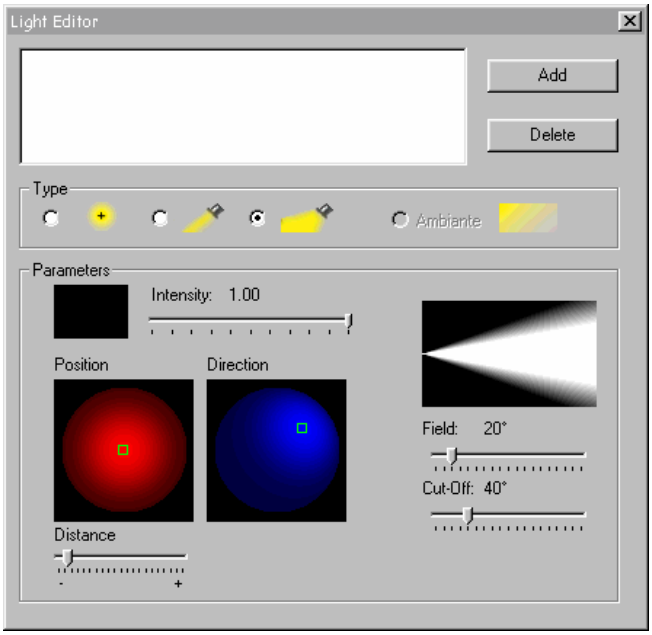


Rendering in TypeView

Light editor in TypeView

You can edit the rendering light, that is to say you can fully parameterize the direction, the source, the intensity, etc. of the rendered object.

Click therefore on the **Light Editor** icon in the upper part of the **TypeView** dialog box. The corresponding dialog box appears on the screen.



In the **Type** part of the dialog box, select the desired option allowing you to define a certain number of settings in the **Parameters** field of the same dialog box.

The ambient light is set by default. From left to right, you can also chose an OMNI light (multidirectional), a beam (parallel directional light) and a spotlight.
In the **Type** section, select the desired light allowing you to define a certain number of settings in the **Parameters** section, such as:

- color (the intensity is defined by a slider)
- position (moving the small square with the mouse)
- direction (idem)
- distance to the object (defined by a slider)
- field and cut-off of the beam (spotlight only).

Camera sight in TypeView

TypeView allows to change the camera sight for any rendered TypeArt object.

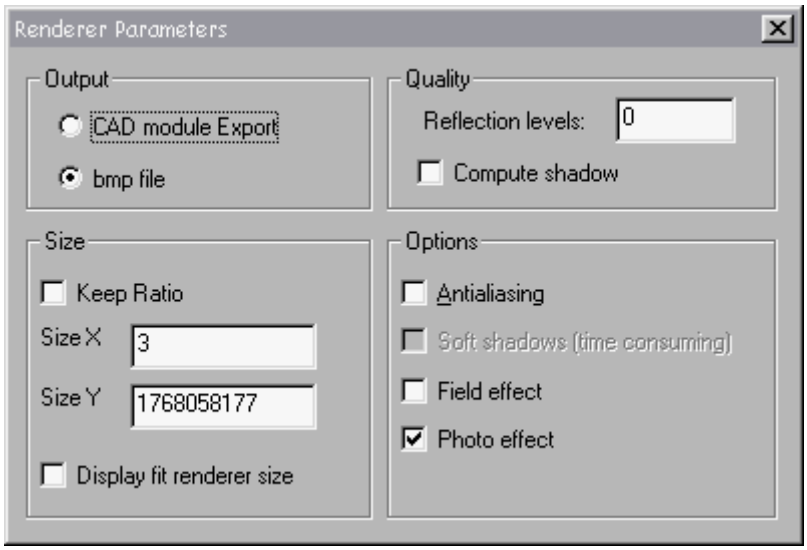
Click therefore on the **Camera sight** icon in the upper part of the **TypeView** dialog box. The **Camera sight** dialog box appears on the screen.

Place your cursor on the green square and push the left mouse button. Keep it pushed and drag your cursor until you reach the desired position. Release then the left button of your mouse.

Renderer parameters

The TypeView module allows you to modify the Renderer parameters.

Click on the **Renderer parameters** icon in the upper part of the **TypeView** dialog box. The **Renderer parameters** appears on the screen.



- **Output** Section

By default the image is generated for the CAD module. It can then be saved with the bitmap editor.
If you click on the **bmp file** radio button, the image is saved in the directory of your choice in bitmap format as soon as the rendering is finished.
You must enter the resolution parameters of the file: number of pixels and number of colors. See the notes in the Size section.

- **Quality** Section

According to your choices in the material editor (color, specular light, transparency, reflection, etc.), these parameters will change completely the way the image is calculated, with more or less reflection and shadowing effects. We advise you to experiment as much as possible the various parameters in order to understand better their action on your drawing.

Values for computing the levels of reflection:
1 = no reflection on the material is calculated,
2 = a reflection on the material is calculated on opaque objects (if, in the material editor, you have a value between 0 and 1 for Reflection),
3 = a reflection on the material is calculated on opaque and transparent objects (if, in the material editor, you have a value between 0 and 1 for Reflection and for Transparency),
4 = calculates a transparent TypeArt object also with these same conditions for material assignment (transparency) and a simple surface (one layer),
5 = calculates the transparency of a surface with two layers (or skins), like for a revolution surface for example.
The display of the rendering is much quicker without any reflection calculation.

- **Size** Section

You can modify the size in X and in Y, in a proportional way if needed.

NOTES
The higher the resolution of the Bitmap (number of pixels), the bigger the size of the file.
The default values of 800 x 800 are a wise compromise.
If there is a background image, the size of the Bitmap is the one of the background image.

- **Options** Section

You can define several other effects:
- Anti-aliasing of the image
- Shadow attenuation, that is with the blurred edge of the shadow (available only with the OMNI light).
- Blurred effect on a part of the composition, like with photo lens.
- General blurred effect.

Rendering in TypeView

Click on the **Render** icon in the upper part of the **TypeView** dialog box to launch the rendering.

APPENDICES

Appendix 1: Importation tips

How can you use a lay out (or a text) created on another computer with another software?

Ex: on Apple Macintosh with Adobe Illustrator

The import of a file from another software can be done if some precautions are respected.

Before exporting your lay out, you have to convert it into curves (especially the text part).
It is a usual function of graphic software.

When a text has been converted into curves, you cannot modify it anymore in the text mode, but the geometrical manipulations can still be executed (size, deformation).
Choice of the format: PostScript, HPGL, DXF, IGES, or ISO.

HPGL (*.HPG)

Created to drive plotters, this language is well known in CAD and in mechanics world in particular. You will find it in most of the existing software. This is a useful solution to import different contours.
However it generates a large quantity of vectors, which can be an inconvenient in some case, especially concerning the quality of the restored curves.

PostScript (*.EPS)

PostScript is the natural format for the description of contours, whether they are fonts or drawings. Contrary to the HPGL which vectorized curves, PostScript defines all the profiles in Bezier curves so there is a better quality at the time of the restoration.
Computation is faster, and the treatment will be optimized because few elements have to be analyzed.
As for the HPGL, you will need to vectorize the composition before exporting it to the PostScript format.

DXF (*.DXF)

This is the Autocad format. Often, contours issued from this format are opened. To avoid to close each contour manually, the Auto-connect function to connect the contours is available in the shaping mode.

REMARK

The quality of the engraving depends on the quality of your composition. The created engraving will be the accurate mirror of the composition supplied. The smallest detail will be reproduced. This performance will allow you mostly to engrave a drawing with delicacy. Sometimes this accuracy will be the origin of imperfections.

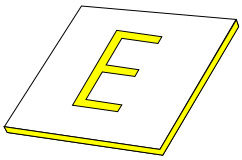
A drawing which has not been created correctly, will produce a mediocre engraving.
Whatever the lines are, straight or curve, they must be as smooth as possible.

Appendix 2: Choices for engraving: recessed or raised

The engraving choice, recessed or raised, is done at the composition and is very simple: each contour is a barrier which delimits the material.

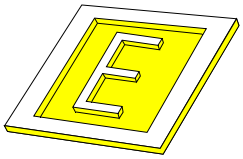
The software will interpret it as follows:
no material - material (1st contour) - no material - material (2nd overlapping contour), etc.

Composition



Result: recessed engraving

Composition



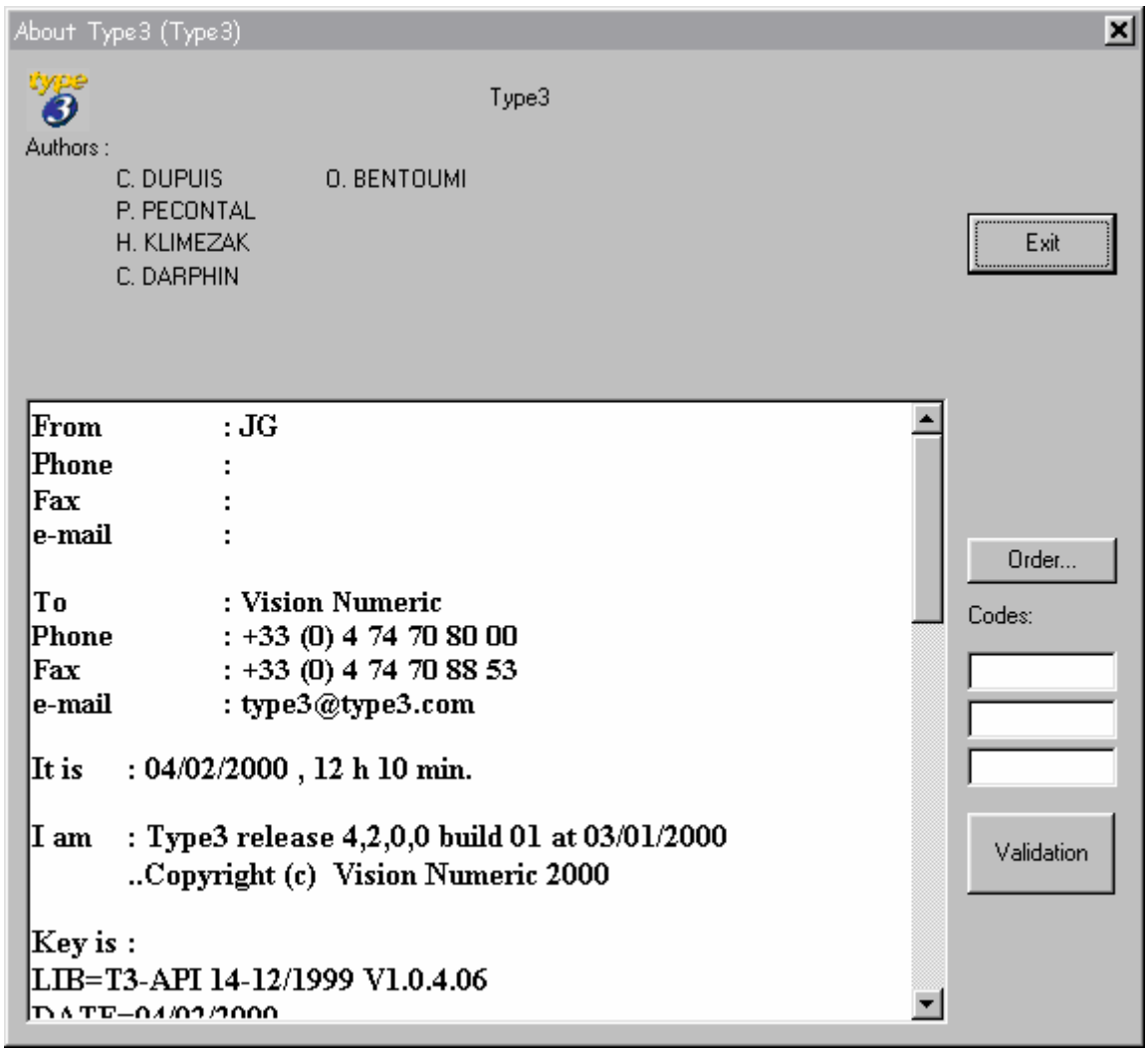
Result: engraving in relief

You just have to add a contour around the composition to create an engraving in relief.

Appendix 3: Order

Should you need to contact your dealer, or should you wish to acquire an additional option, please send by fax the details of your dongle after filling in your coordinates. To obtain details of your dongle, please follow the procedure hereafter.

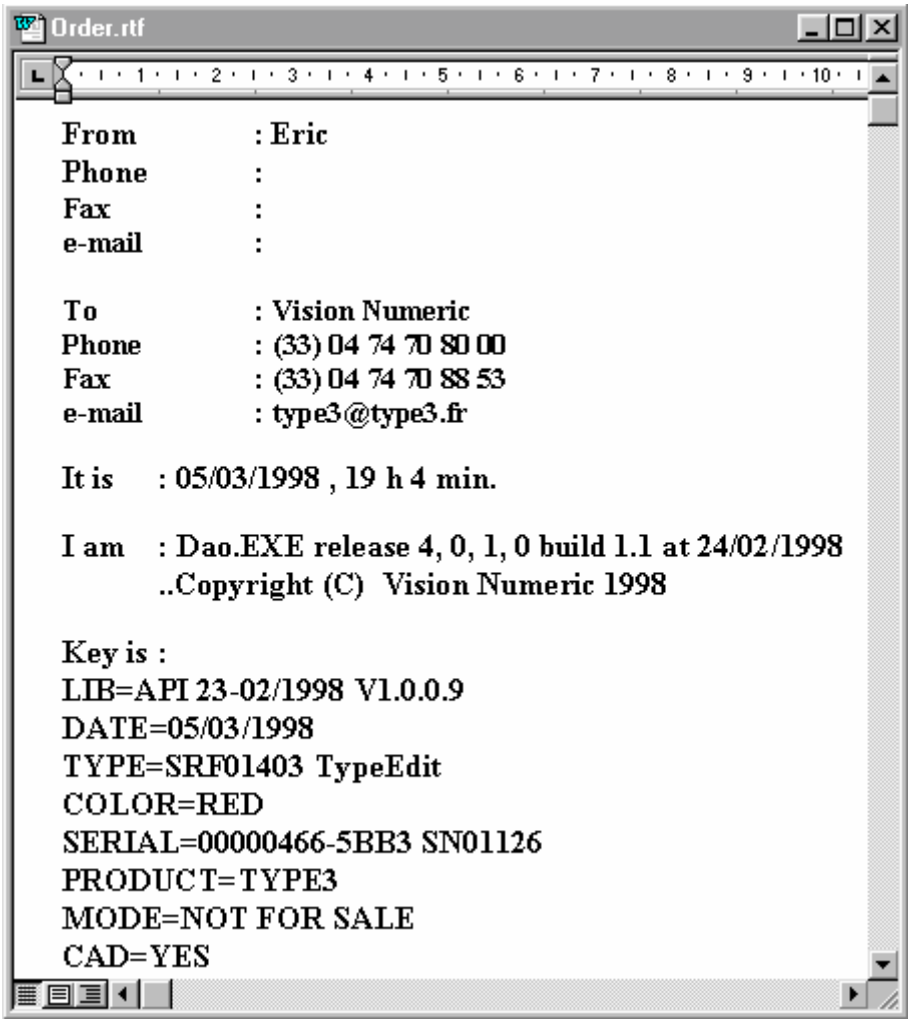
Click in the menu on the icon .



Be careful not to modify the code numbers in this window!!!

Then select the **Order** command.

Enter the following information in the text editor displayed on the screen: address, telephone, fax and E-mail in the fields **From**, **Phone**, **Fax** and **E-mail**.



Then print the note-pad with **File → Print**, and fax it! **Don't forget to mention clearly the nature of your problem!!!**

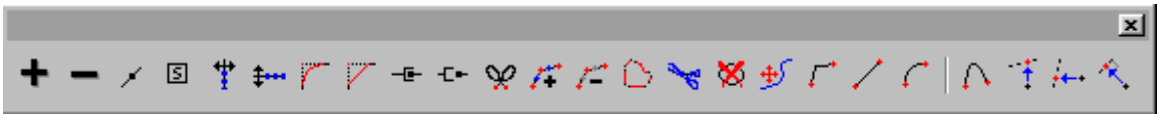
You will receive in return the sequence(s) allowing the activation of the ordered option(s).
Each sequence must be entered in the **Code** field of the dialog box **About** and must be validated by clicking on the button **Validation**.
The display of the message **Success** proves that this task went well. Do the operation as many times as the number of given sequences.

To quit this screen, click on or on **OK**.

Appendix 4: Tool box

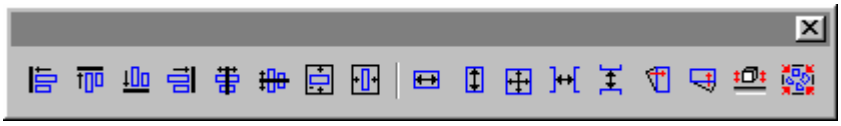
Each tool available in the software is represented by an icon symbolizing its function.
The icons defined with the mention **Tools** and marked with a little arrow on the bottom right corner are launching a new box with icons described on the following page.
When the function is available, the icon becomes darker.

The shaping tools



Geometrical shapes tools

Alignment tools

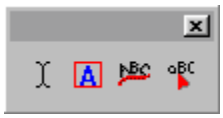


Duplication tools

Transformation tools

Special effects tools

Text mode



Surface tools

Measuring tools

TypeArt tools

The icon shown above allows you to switch to the **TypeArt** module.

Permanent toolbar of the **TypeArt** module

Click on the icon to display the **Transformation tools** of the TypeArt module

Click on the icon to display the **Duplication tools** of the TypeArt module

Click on the icon to display the **Special effects tools** of the TypeArt module

Click on the icon to display the **CAM tools** of the TypeArt module

click on the icon to display the **Visualization tools** of the TypeArt module

Appendix 5: Tool bar

Appendix 6: Shortcuts

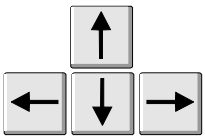
The Function keys

F1: Help
F2: Numerical coordinates
F3: Snap mode
F4: Preview
F10: Options

The tabulation key

Automatic centering of text: justification left/right/center

The arrow keys



Accurate move of selected item. You can modify their setup: therefore see the section **Arrow move** in the chapter **Configuration**.

The Ctrl key

Ctrl + **1**: Front view on the **XY** axes
Ctrl + **2**: Cross section view on the **YZ** axes
Ctrl + **3**: Cross section view on the **XZ** axes
Ctrl + **4**: **3D Engraver** view on the **XYZ** axes
Ctrl + **5**: **3D Iso** view on the **XYZ** axes
Ctrl + **6**: **Free 3D** view on the **XYZ** axes through numerical values (displays dialog box **View point**).

Ctrl + **N**: New file
Ctrl + **O**: Open
Ctrl + **S**: Save
Ctrl + **M**: Open model
Ctrl + **Z**: Undo
Ctrl + **A**: Redo
Ctrl + **X**: Cut
Ctrl + **C**: Copy
Ctrl + **V**: Paste
Ctrl + **L**: Select all
Ctrl + **W**: Redraw
Ctrl + **R**: Fast redraw
Ctrl + **Shift key**: When drawing an object, it is centered

Ctrl with **Sketching tool**: Angular point
Ctrl with **Rectangle tool**: Square
Ctrl with **Ellipse tool**: Circle
Ctrl with **Selection tool**: Multiple selection/Deselection
Ctrl with **Duplication tool**: Move the copy on one axis
Ctrl with **Mirror tool**: Copy + Mirror
Ctrl with **Scaling tool**: Scale 2, 3.
Ctrl with **Rotation**: 15° steps
Ctrl + : Vectorizing parameters
Ctrl + : Edition and creation of fonts
Ctrl + : Choice of the kind of marker

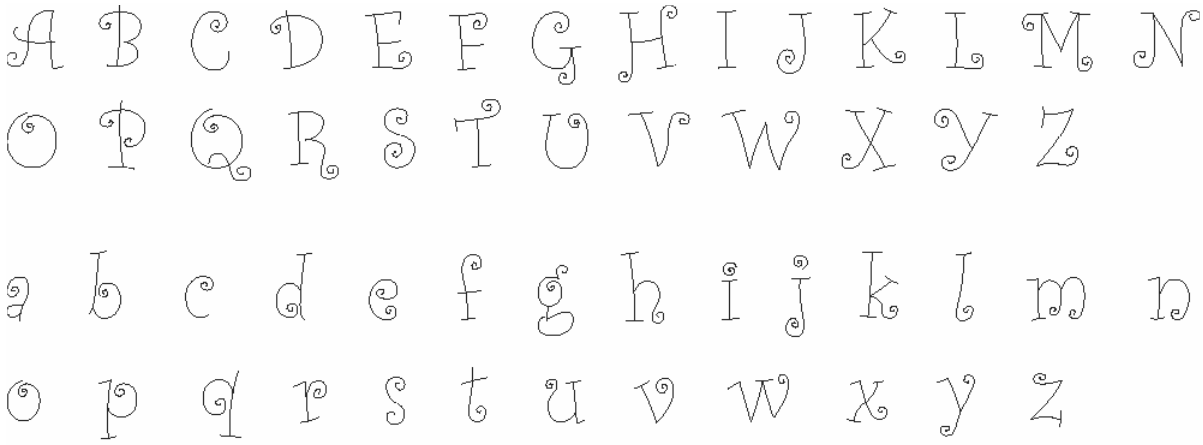
The Shift key

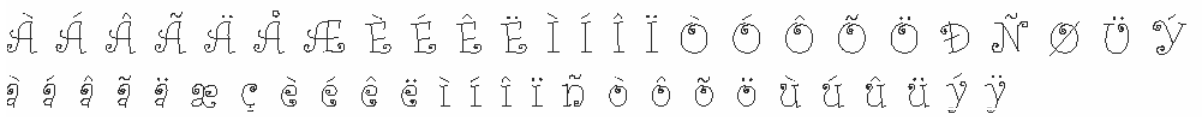
Shift with **Curves shaping mode**: Tangential point
Shift with **Ellipse tool**: The ellipse is centered on the first selected point
Shift with **Polygon tool**: Two opposite points
Shift with **Star tool**: two opposite points with circle
Shift with **Circle**: Clockwise direction
Shift with **Arc by 3 points**: Third point defined between first and second
Shift with **Copy/Paste function**: Copy of selected item
Shift with **Scaling with mouse**: Symmetrical action

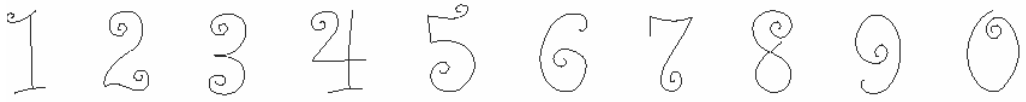
Appendix 7: Vision numeric fonts

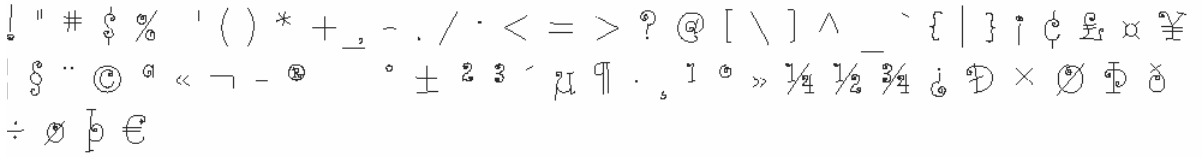
- **VN Curlz**

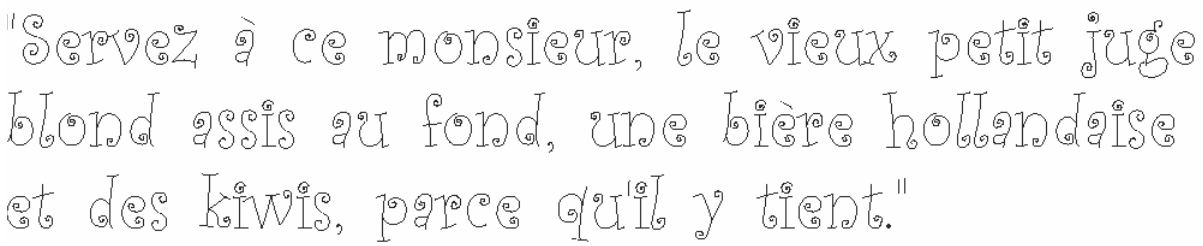
This font is made of 190 characters.











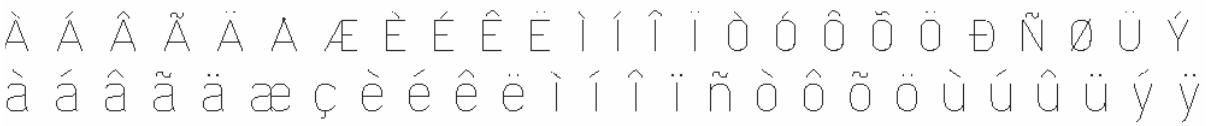
- **Din Filaire**

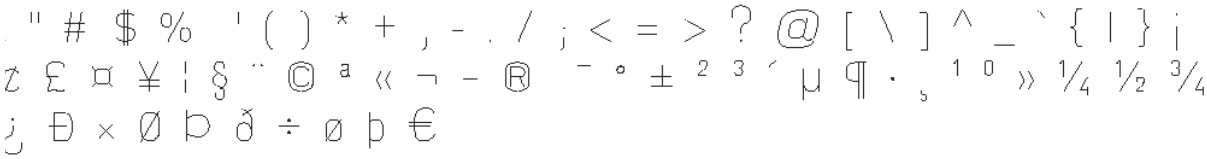
This font is made of 190 characters and 324 kerning pairs.

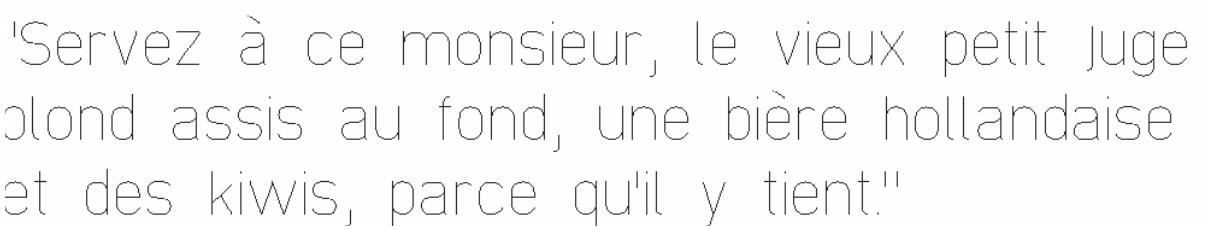
A B C D E F G H I J K L M N
O P Q R S T U V W X Y Z
a b c d e f g h i j k l m n
o p q r s t u v w x y z
À Á Â Ã Ä Å Æ È É Ê Ë Ì Í Î Ï Ò Ó Ô Õ Ö Ð Ñ Ø Ü Ý
à á â ã ä æ ç è é ê ë ì í î ï ñ ò ô õ ö ù ú û ü ý ÿ
1 2 3 4 5 6 7 8 9 0
! " # $ % ' ( ) * + , - . / ; < = > ? @ [\ ] ^ _ ` { | } ¡ ¢ £
¤ ¥ ¦ § ¨ © ª « ¬ – ® ¯ ° ± ² ³ ´ µ ¶ · ¸ ¹ º » ¼ ½ ¾ ¿ Ð ×
Ø Þ ð ÷ ø þ €
"Servez à ce monsieur, le vieux petit juge blond assis au fond, une bière hollandaise et des kiwis, parce qu'il y tient."

- **VN Glass**

This font is made of 190 characters and 110 kerning pairs.







- **Vn Helvetica 1L**

This font is made of 190 characters and 109 kerning pairs.

A B C D E F G H I J K L M N
O P Q R S T U V W X Y Z
a b c d e f g h i j k l m n
o p q r s t u v w x y z
À Á Â Ã Ä Å Æ È É Ê Ë Ì Í Î Ï Ò Ó Ô Õ Ö Ð Ñ Ø Ü Ý
à á â ã ä æ ç è é ê ë ì í î ï ñ ò ô õ ö ù ú û ü ý ÿ
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"Servez à ce monsieur, le vieux petit juge
blond assis au fond, une bière hollandaise
et des kiwis, parce qu'il y tient."

- **VN Script**

This font is made of 190 characters and 256 kerning pairs.

- **VN Serif**

This font is made of 190 characters and 380 kerning pairs.

A B C D E F G H I J K L M N
O P Q R S T U V W X Y Z
a b c d e f g h i j k l m n
o p q r s t u v w x y z
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